U S E R ' S M A N U A L

WMS

Watershed Modeling System

BOSS INTERNATIONAL, INC. and
BRIGHAM YOUNG UNIVERSITY
© Copyright 2000 ▪ All Rights Reserved

WMS Installation Guide for Personal Computers

This section describes the system requirements, installation procedures, and start-up of WMS for Microsoft Windows 3.x, Microsoft Windows for Workgroups 3.11, Microsoft Windows 95, and Microsoft Windows NT on a personal computer.

Inventory

If you downloaded the software from the internet, then the following inventory section does not apply. However, if you received the shipment from BOSS International, then included with your distribution of WMS should be the following items:

- CD-ROM containing the WMS executable and resource files, as well as the various modules that you may have purchased. These modules include Map, TIN, DEM, HEC-1, TR20, TR55, Rational, NFF, and other interface modules.
- WMS License Diskette (Serial Numbered Diskette).
- WMS Hardware Lock (if a hardware locked version was purchased).
- WMS User Manual.
- This installation guide.
- Technical documentation for any model interfaces you may have ordered (optional).

If you are missing any of the above items, please contact us.

How to Contact BOSS International

BOSS International can be reached at:

BOSS International
6612 Mineral Point Road
Madison, WI 53705
USA

Monday through Friday 8:00am to 5:00pm CST (6hrs behind GMT)
Telephone: 608-258-9910

24 hours/7days a week
Fax: 608-258-9943
BBS: 608-238-5266
Email: support@bossintl.com
WWW Site: http://www.bossintl.com

System Requirements

This program requires the following to functionally operate:

- A system running Microsoft Windows NT, Windows 95, Windows for Workgroups 3.11, or Windows 3.1.
- A 486 or larger micro-processor (486sx processors are not supported).
- 16 MB of extended memory (minimum). However, 32 MB is recommended.

Program Installation

WMS is installed onto your PC using an easy to use installation program. This section discusses how the installation program installs WMS application onto your computer's hard drive. Once the program installation is completed, the program license must be installed. The program license can either be installed automatically using the installation program (as discussed in this section), or manually as discussed in a following section titled *Manually Licensing WMS*.

How to install this software:

1. Insert the supplied CD-ROM into your computer CD-ROM drive.

2. If you are running Windows 95 or NT 4.0 (or newer operating system) the installation program should startup automatically when this CD-ROM is inserted.

   If you are running Windows 3.x, Windows NT 3.5 or 3.51, then using File Manager double click on the SETUP.EXE file contained in the root directory of the CD-ROM disk.

3. Select a product to install and then click [OK].

   Note that some installation notes may be displayed. This information informs you what the installation program will be doing when it installs the application onto your computer hard drive.

4. A dialog box will be displayed allowing you to define a destination drive and directory for installing the software. Once you have defined a drive and directory, click [OK].

   Note that installing the software may take some time. A progress bar will be displayed, showing you the progress of the installation.

5. Once the software has been installed, a dialog box will be displayed asking you to install the program license.

   If you received a *hardware lock* with your shipment, the installation program will explain how to install the hardware lock to enable the program.

   If you received a *serial numbered license diskette* with your shipment, then the installation program will explain how to install the software license to enable to program.

Starting Up WMS

Once the installation is complete, a new program group will be created on your Windows desktop with the WMS icon contained within it. Once you have verified that installation is complete, you may start up WMS by double clicking on the WMS icon.

Troubleshooting Microsoft WIN32s

WMS is a 32-bit application that utilizes WIN32s. WIN32s is a set of dynamic link libraries (DLLs) written by Microsoft that allow 32-bit Windows applications to run transparently under Windows 3.1 and Windows for Workgroups 3.11. Part of the WMS installation installs WIN32s onto your hard drive. If you already have WIN32s properly installed, this procedure will automatically be skipped. For more information on WIN32s, please contact Microsoft.

The sections below describe some possible solutions if problems are encountered when installing WIN32s.

Potential WIN32s Error Messages:

- File Error: Cannot find OLECLI.DLL.
- WIN32S-Error: Improper installation. WIN32s requires WIN32.EXE and WIN32S16.DLL to run. Reinstall WIN32s.
- WIN32s-Error: Improper installation. Windows requires W32s.486 in order to run. Reinstall WIN32s.

Recommended Solutions:

- If you are having video problems, check to see if you have an S3 video card. Certain S3 drivers do not work with WIN32s. Use either the generic drivers shipped with Windows or contact your video card manufacturer for an updated driver.
- For additional information on the S3 driver and WIN32s, please see the following article(s) in the Microsoft Knowledge Base:

  ARTICLE-ID: Q117153
  Title: PRB: Display problems with
  WIN32s and the S3 driver.

- Make sure the following line is in your SYSTEM.INI file:

  ```
  DEVICE=*VMCPD
  ```

- If you have a printer driver by Lasermaster™, delete it or comment it out, as it interferes with installing WIN32s. Once WIN32s is installed, you can then re-enable the printer driver.
- Try reinstalling WIN32s.
- Make sure paging is enabled. From the Control Panel, select the *386 Enhanced icon*, choose *Virtual Memory*, then choose *Change*. Verify that the drive type is NOT set to none. The drive type may be set to either "temporary" or "permanent." “Permanent” is recommended.

Disabling Microsoft WIN32s

It should not be necessary to disable WIN32s. The WIN32s DLLs will only be loaded when a WIN32s application is executed. The WIN32s VxD is loaded when Windows starts, but has little memory overhead. If you must disable WIN32s or wish to do a clean reinstallation of WIN32s, take the following actions:

1. Remove the WIN32s VxD line from Windows SYSTEM.INI files in the [ENH386] section:

   DEVICE=C:\WINDOWS\SYSTEM\WIN32S.386

2. Delete the W32SYS.DLL and the WIN32S16.DLL files from the <WINDOWS>\SYSTEM directory and all files in the <WINDOWS>\SYSTEM\WIN32S subdirectory. (Note that <WINDOWS> is the Windows installation directory such as C:\WINDOWS.)

3. Restart Windows.

Manually Licensing WMS

The installation program copies WMS and various other files related to application onto your hard drive. However, without enabling the program license using either the software license diskette, password license, a hardware lock, or a network hardware lock, the software will only operate in *demo mode*—not allowing you to print or save.

The following sections discuss how to manually install the program license.

Software License

If you were furnished with a License Diskette with your shipment, use this method for installing the program license.

Note that this method only works for computers operating with Windows 3.x and Windows 95. If you are using another operating system, such as Windows NT, you will need to use an alternate licensing method.

1. Place the License Diskette into your computer hard drive.

2. Using Windows Explorer or Windows File Manager, run the LICENSE.EXE application contained on the License Disk.

3. The LICENSE software license program will prompt you completely through the software license process.

If you need to uninstall the software license from the computer it is installed on, the LICENSE program can again be used.

Password License

The password license method can be used to enable the software on a computer that the software license does not work with (e.g., Windows NT) in which a hardware lock has not been purchased.

The first time you bring up WMS, a message telling you that you are running an *Evaluation Copy* will be displayed. This message will appear anytime WMS is run on a computer that does not have a program license.

To install a password license, follow these steps:

1. Start up WMS.

2. Select the *Register* command from the File Menu. The Register dialog box will appear.

   The Register dialog box will list a program security string that is unique for the computer it is to be installed on. It also contains a password edit field. If this is the first time you are registering the program, this field will be empty. If the computer you are running on is already registered, the current password will be displayed in the field. In addition, the enable status of each of the WMS modules will be listed in the dialog.

   Click on [Details] to see detailed description of the security string.

   To register your computer with a WMS password license, contact BOSS International Technical Support and provide them with the security string. BOSS International Technical Support will provide you with a password that will license the program for your computer.

3. Enter the password into the password edit field provided.

4. Click on [Register] to register the password license information.

If the password was entered correctly, the enable status of the modules will be updated below in the dialog box. If the status did not update correctly, double check that the password was correctly entered or that the security string was properly read.

Once WMS has been registered, a file called `WMSPASS.TXT` will be created in the Windows System directory. This file was created for storing the above password and contains only the password. If you are running on a network, it will be saved in the Windows System directory of the currently active Microsoft Windows.

Hardware Lock License

If you were furnished with a hardware lock with your shipment, this section will discuss how to enable the hardware lock license.

Under most operating systems, you must install a hardware lock driver—a small, memory-resident program—first, so that your computer can recognize the presence of a hardware lock. Using the supplied CD-ROM installation program, from the Utilities Menu you can select to install the hardware lock driver. The installation program will automatically install the appropriate hardware lock driver for the operating system your computer has.

Troubleshooting Hint

If you are experiencing difficulties, you may wish to run the diagnostic program that is available from the CD-ROM installation program's Troubleshooting Menu. This program will check to see if the printer port has been located correctly and whether the hardware lock is working properly.

When running this utility, answer `Y` ("yes") to all prompts. The hardware lock is working properly if the program responds with the statement, "`1 SentinelSuperPro key detected !`"

If, after running the diagnostic program you continue to experience difficulties, please feel free to contact our Technical Support staff.

Network Hardware Lock License

The "network" hardware-locked version enables WMS to be installed on a network server, thus allowing multiple users to use the program from separate computers on the network.

Under most operating systems, you must install a hardware lock driver—a small, memory-resident program—first, so that your computer can recognize the presence of a hardware lock. Using the supplied CD-ROM installation program, from the Utilities Menu you can select to install the hardware lock driver. The installation program will automatically install the appropriate hardware lock driver for the operating system your computer has.

You will also need to install a security server driver on the network file server. The security server is responsible for keeping track of the number of users using WMS at any one time. Using the supplied CD-ROM installation program, from the Utilities Menu you can select to install the security server driver. The installation program will automatically install the appropriate driver for the operating system your network server has.

Although the network hardware-locked version of WMS can be installed on any computer on your network, we strongly suggest that it be installed on your network file server.

Troubleshooting Hint

If you are experiencing difficulties, you may wish to run the diagnostic program that is available from the CD-ROM installation program's Troubleshooting Menu. This program will check to see if the network server printer port has been located correctly and whether the network hardware lock is working properly.

When running this utility, answer `Y` ("yes") to all prompts. The hardware lock is working properly if the program responds with the statement, "`1 SentinelSuperPro key detected !`"

If, after running the diagnostic program you continue to experience difficulties, please feel free to contact our Technical Support staff.

Using WMS on Other Computers

WMS is licensed on a single user basis. This software can be installed on only one computer for use by one user at a time. (One "extra" installation is provided for backup purposes *only* with the software license diskette.) If this software is to be installed on multiple computers, each computer requires a separate license.

To use the program on a different computer, you will first need to uninstall WMS. An *Uninstall* icon was created in the application's group during installation. Double-click the *Uninstall* icon to begin the uninstall process.

If you are using the software license method, you will also need to manually uninstall the software license using the LICENSE program contained on the License diskette. The LICENSE program will prompt you through how to uninstall the software license.

The LICENSE licensing program will query you to confirm that you want to move the program license. If you have any difficulties, please contact BOSS International Technical Support for assistance.

Network Server Technical Information

The Network Server Version of WMS allows you to install the software on a network file server, enabling anyone connected to the file server to access the software without having the software actually installed on their individual computer.

The Network Server Version (by default) allows only one concurrent network user to use the software at any time. Additional concurrent network user counts may be purchased from BOSS International, allowing multiple users to access the software simultaneously from the file server.

For WMS to properly prevent accidental file overwriting, a file-sharing, file-locking program (i.e., SHARE.EXE when using DOS or VSHARE.386 when using Windows for Workgroups) must be loaded on both the file server and the individual workstation. For more information on these programs, consult your DOS or Windows for Workgroups User Manuals.

Troubleshooting

The following is a list of common errors which may be encountered upon installing and running WMS.

- **Not running on a 386 DPMI implementation**.

  WMS is a 32-bit Windows application. In order to run WMS, Windows must be running in "Enhanced Mode." To check which mode Windows is running in, open the "About Program Manager" menu item from the Help Menu. The Windows Mode will be displayed here. If you are running in Standard Mode, exit Windows and restart with the following command:

  ```
  WIN/3
  ```

- **Not enough memory for Application.**

  WMS requires a minimum of 16 Mbytes of RAM to run effectively. You may require additional memory, depending on the models you intend to create. In order to obtain more memory, there are a few things you can do:

  1. Exit other programs currently running.
  2. Edit your `CONFIG.SYS` file and turn off any RAM drives.
  3. From the control panel choose the 386 Enhanced icon. Select the virtual memory option and increase the size of your virtual memory.

- **WMS dialog boxes appear to take up the entire screen and some will be larger than the screen.**

  Check to make sure your monitor is running in Super VGA display mode. WMS requires a display minimum screen resolution of 1024x768. Use the Windows setup command to change the display mode. Check with your Microsoft Windows documentation for a description of how to change the screen display resolution.

- **WMS immediately quits upon starting up.**

  It is possible that WIN32s was not installed successfully. Try running the FreeCell program that was installed on you hard drive as part of the WIN32s installation. If it does not run, try reinstalling WIN32s. If the problem persists, there may be an incompatibility between your systems hardware and Microsoft's WIN32s. Contact the manufacturer of your computer.

- **You used the alternative method to register the licensing of WMS (if you are running Windows NT) and WMS comes up as an evaluation copy, even though WMS has been previously registered.**

  WMS will startup as an Evaluation Copy only if the WMS password file does not exist or an incorrect password is defined in the file. Check that the file `WMSPASS.TXT` exists in the Windows System directory of the Windows program currently in use. WMS must be run from the same Windows program and hard drive that the program was originally registered for. If the `WMSPASS.TXT` file has been deleted, or you intend to use WMS on another computer or hard drive, contact our technical support staff for an updated password. No additional fee will be charged, provided that WMS will be run on only one computer at a time. Passwords for additional computers will be subject to additional licensing fees.

- **The application continues to run as an Evaluation Copy (Windows 3.1 or Windows 95 only).**

  If the application starts up as an Evaluation Copy even after running EVMOVE, you may need to load the driver `AZMEMORY.DRV` (located in the `\WMS` directory on your computer), to reserve a small amount of DOS conventional memory when Windows is run.

  To install the `AZMEMORY.DRV` driver, complete the following steps:

  1. Copy the `AZMEMORY.DRV` file from your `\WMS` directory to your `\WINDOWS\SYSTEM` subdirectory.

2. Edit your SYSTEM.INI file (located in your \WINDOWS directory) to add the following entries:

   In the [boot] section of SYSTEM.INI, edit the "drivers" line to include AZMEMORY.DRV. If there are other entries on this line, the AZMEMORY driver should be first. This will make sure the driver is loaded first. After editing the line, it should look something like this:

   drivers=azmemory.drv mmsystem.dll

   In the [drivers] section of SYSTEM.INI, add the line:

   AzMemory=azmemory.drv

3. Save the SYSTEM.INI after the specified changes have been made. You must then restart Windows for the AZMEMORY.DRV to be loaded.

WMS Installation Guide for UNIX Workstations

This section describes the system requirements, installation, and start-up of WMS for UNIX workstations.

Inventory

Included with your distribution of WMS should be the following items:

- Magnetic tape containing the WMS executable, its resource files, and any groundwater models ordered with this distribution (unless the software was received via the Internet).
- WMS Reference Manual.
- This installation guide.
- Technical documentation for any model interfaces you may have ordered (optional).

If you are missing any of the above items, please contact us.

How to Contact BOSS International

BOSS International can be reached at:

BOSS International
6612 Mineral Point Road
Madison, WI 53705
USA

Monday through Friday 8:00am to 5:00pm CST (6hrs behind GMT)
Telephone: 608-258-9910

24 hours/7days a week
Fax: 608-258-9943
BBS: 608-238-5266
Email: support@bossintl.com
WWW Site: http://www.bossintl.com

Installation

WMS is installed on a UNIX workstation using a two-step process. In the first step, WMS is either copied from the magnetic tape or transferred via the Internet to your computer's hard drive. During the second step, the authorization files are installed on to the hard drive, such that WMS can be run.

Downloading WMS

You may obtain WMS via the Internet, either from our FTP site (ftp.bossintl.com) or our World Wide Web site (http://www.bossintl.com). Please contact BOSS International Technical Support for the current file name to complete this process.

Once the program is placed into the current directory of your hard drive, you should verify that a directory named WMS now resides on your hard drive. The following files and directories should be contained in this directory:

```
COLORS.BYU
FONT.INC
WMS
TUTORIAL/
```

Depending on which modules you have purchased, other directories also may be present.

Copying From the Tape

If you wish to install WMS on a system area on your computer so that all users may have access to the software, you should log in as the super user (root). If you are installing it in a particular user's area and intend only for that user to use the program, you do not have to be logged in as the super user. Once you are logged in, change directories to where WMS is to be installed. You do not need to create a new directory as the installation process will automatically create the directory WMS in which all files will be placed.

Insert the tape into the tape drive. Type the following command to copy the contents of the tape into the current directory of your hard drive:

```
TAR - XV
```

Once the contents of the tape have completed being copied onto the current directory of your hard drive, you should verify that a

directory named WMS now resides on your hard drive. The following files and directories should be contained in this directory:

```
COLORS.BYU
FONT.INC
WMS
TUTORIAL/
```

Depending on which modules you have purchased, other directories also may have been copied.

Registering WMS

The first time you bring up WMS, a message telling you that you are running an Evaluation Copy will be displayed. Any time WMS is run on a computer that has not been *registered* to run the program, this message will appear. WMS can be run in this mode, but the printing and saving of files will be disabled. To register your computer to run WMS, the *Register* command is used.

1. Start up WMS.

2. Select the *Register* command from the File Menu. The Register dialog box will appear.

   The Register dialog box will list a program security string that is unique for the computer it is installed on. It also contains a password edit field. If this is the first time you are registering the program, this field will be empty. If the computer you are running on is already registered, the current password will be displayed in the field. In addition, the enable status of each of the WMS modules will be listed in the dialog. Click on [Details] to see detailed description of the security string.

   To register your computer with a WMS password license, contact BOSS International Technical Support and provide them with the security string listed at the top of the dialog box. BOSS International Technical Support will provide you with a password that will license the program for your computer.

3. Enter the password into the password edit field provided.

4. Click on [Register] to register the password license information.

If the password was entered correctly, the enable status of the modules will be updated below in the dialog box. If the status did not update correctly, double check that the password was correctly entered or that the security string was properly read.

Once WMS has been registered, a file called `.WMSPASS` will be created in the current directory. If this is the same directory from which you

will always be running WMS, the .WMSPASS file should remain there, otherwise you may want to move the file to a resources directory.

Running WMS

If you have installed WMS in the system area, or added the WMS directory to the path, WMS can be executed by typing *WMS* from any directory.

If you have installed WMS for a single user, then you should change directories to where the WMS is located and type *WMS*. If you do not wish to change to this directory every time WMS is to be executed, you may wish to create an alias, use a script file, or change the path in your .LOGIN file.

Since all tutorial files are found in the tutorial directory, you may need to set protections so that all users can read and use them.

Using WMS on Other Computers

WMS is licensed on a single user basis. This software can be installed on only one computer for use by one user at a time. If this software is to be installed on multiple computers, each computer requires a separate license. You may purchase additional licenses for WMS by contacting BOSS International.

If it becomes necessary to move WMS from the currently registered computer to another, contact BOSS International Technical Support for an updated password. No additional fee will be charged, provided WMS is only run on one computer at a time.

Technical Notes

The following is a list of technical notes to assist you in the operation of WMS.

The WMS Resources Directory

The default installation of WMS will place the resource files in the WMS directory along with the executable. The resource files include the following:

```
COLORS.BYU
FONT.INC
WMS.INI
.WMSPASS
```

If WMS will not be run from the installation directory, it will be necessary to specify the path name to where the resource files are located. Upon starting up WMS, an argument may be added to specify the location of these files. For example, if the resource files are located in a directory named *Resources* paralleled to where WMS will be run, you would enter the following:

```
WMS  ../RESOURCES
```

When WMS starts up, it will first look for the resource files in the specified resources directory. If they are not found there, it will look for them in the current working directory. If the resource files still are not located, WMS will use the default values for the information contained in the resource files.

Installation Note
If the `.WMSPASS` file is not found, WMS will start up as an Evaluation Copy.

The WMS Script File

It is often more convenient to use the file to launch the program than to type the path name to the WMS executable and resources directory. The script file contains the exact command line that you would use to launch the program. For example, the WMS executable and resource files may be located in a directory named /APPS/WMS. To create a script file to launch the WMS application in that directory, you would do the following:

- Create a file in the directory that you intend to run WMS from (or in the path specified in your .CSHRC file) using a text editor. The name of the file will be the command you will use to launch WMS. The following line would be entered in the file:

```
APPS/WMS/WMS  /APPS/WMS
```

- The file /APPS/WMS/WMS is the WMS executable and the path /APPS/WMS represents the path name to the resources directory. In this case, both WMS and the resources are located in the same directory.

- After saving the file, execute privileges will need to be given to the script file. If the file name is WMS, you would enter this name at the UNIX command line.

Customizing the WMS Colors

The colors used by the WMS interface are defined in the resource file COLOR.BYU. This file defines four sets of colors used by the various widgets within the program. The colors are specified as RGB's. You may want to make a back-up of this file before making any modifications. Similar to the FONTS.INC resource file, if the COLORS.BYU file is not found, WMS will select a set of default colors.

Troubleshooting

The following is a list of common problems which may be encountered upon installing or running WMS.

- **Your system does not respond to the command TAR -XV**.

  It may be necessary to specify the device name by using the following variation of the TAR command:

  ```
  TAR -XVF /DEVICENAME
  ```

  Ask your system manager for the device name of the tape drive being used.

- **WMS does not run or the message "*command not found*" appears on your screen**.

  Be sure that the WMS script file has execution (x) privileges. You should also ensure that the directory where WMS or the script file is run from is in your path. The path is defined in the .CSHRC file located in your home directory.

- **The following font and colors file message(s) appear:**

  ```
  Font file missing or contains or bad font -
  loading temp fonts

  Error opening colors resource file
  ```

  The location of the WMS resource files FONTS.INC and COLORS.BYU cannot be determined. In this case, WMS will use default colors and fonts. To utilize the fonts and colors specified in the WMS resource files, the path name to where these files are located should be used as the second argument when launching the program. You may want to refer to the section titled *The WMS Resources Directory*.

  You should also ensure that the user has read privileges on the resource files contained in the resources directory and that the path specified in the script file is the correct path to the directory.

- **WMS comes up as an Evaluation Copy, even though you have registered WMS for this computer.**

  WMS will come up as an Evaluation Copy only if the WMS password file does not exist or an incorrect password is defined in the file. Check that the file `.WMSPASS` exists in either the directory you started WMS from (the current working directory) or the resources directory. You may want to refer to the section titled *The WMS Resources Directory*.

You should also ensure that the .WMSPASS file has read privileges for the user that is running WMS. WMS must be run from the same computer that the program was originally registered for. If the .WMSPASS file has been deleted or you intend to use WMS on another computer, please contact BOSS International for an updated password. No additional fee will be charged, provided WMS will be run on only one computer at a time. Passwords for additional computers will be subject to additional licensing fees.

WMS

Watershed Modeling System

REFERENCE MANUAL

WMS 6.0

Copyright © 1999 Brigham Young University - Environmental Modeling Research Laboratory

All Rights Reserved

Unauthorized duplication of the *WMS* software or user's manual is strictly prohibited.

THE BRIGHAM YOUNG UNIVERSITY ENVIRONMENTAL MODELING RESEARCH LABORATORY MAKES NO WARRANTIES EITHER EXPRESS OR IMPLIED REGARDING THE PROGRAM *WMS* AND ITS FITNESS FOR ANY PARTICULAR PURPOSE OR THE VALIDITY OF THE INFORMATION CONTAINED IN THIS USER'S MANUAL

The software *WMS* is a product of the Environmental Modeling Research Laboratory of Brigham Young University.

TABLE OF CONTENTS

1 INTRODUCTION 1-1

1.1 OVERVIEW 1-1
1.1.1 Running WMS for the First Time 1-1
1.1.2 Digital Terrain Modeling 1-2
1.2 FUNDAMENTAL *WMS* MODELING CONCEPTS 1-3
1.2.1 Guidelines for Using Feature Object (GIS) Vector Data 1-3
1.2.2 Guidelines for Using DEM Data 1-6
1.2.3 Guidelines for Using TIN Data 1-15
1.2.4 Guidelines for Using ARC/INFO® and ArcView® Formatted Data 1-21
1.3 MODULES 1-23
1.3.1 Triangulated Irregular Network (TIN) Module 1-23
1.3.2 DEM Module 1-24
1.3.3 Map Module 1-24
1.3.4 Hydrologic Modeling (Tree) Module 1-24
1.3.5 Grid Module 1-25
1.3.6 Scatter Point Module 1-25
1.4 CONSIDERATIONS FOR READING EXISTING *HEC-1* FILES 1-25
1.5 CONSIDERATIONS FOR READING EXISTING *TR-20* FILES 1-26
1.6 CONSIDERATIONS FOR READING EXISTING HSPF FILES 1-27

2 GENERAL TOOLS 2-1

2.1 INTRODUCTION 2-1
2.2 WMS SCREEN 2-1
2.3 GRAPHICS WINDOW 2-2
2.4 HYDROGRAPH WINDOW 2-2
2.5 TOOL PALETTE 2-3
2.5.1 Module Palette 2-3
2.5.2 Static Tool Palette 2-4
2.5.3 Dynamic Tool Palette 2-5
2.5.4 Macros 2-6
2.6 EDIT WINDOW 2-6
2.6.1 Coverage and Data Set Selection 2-6
2.6.2 Coordinate Edit Boxes 2-6
2.6.3 Help Strip 2-6
2.7 MENU BAR 2-7
2.8 FILE MENU 2-7
2.8.1 WMS File Types 2-7
2.8.2 New 2-8
2.8.3 Open 2-8
2.8.4 Save As 2-9
2.8.5 Save Current Settings 2-10
2.8.6 Import File 2-10
2.8.7 Export File 2-20
2.8.8 Print 2-22
2.8.9 Printing Files (UNIX Platforms) 2-25

2.8.10 Printing Files (PC Platforms) *2-25*
2.8.11 Viewing Files *2-25*
2.8.12 Get Info *2-25*
2.8.13 Demo Version *2-25*
2.8.14 Register *2-26*
2.8.15 Exit *2-26*
2.9 EDIT MENU 2-26
2.9.1 Deletion Commands *2-26*
2.9.2 Selection Commands *2-27*
2.9.3 Confirm Deletions *2-27*
2.9.4 Copy to Clipboard (PC Platforms Only) *2-27*
2.10 DISPLAY MENU 2-28
2.10.1 Display Options *2-28*
2.10.2 Contouring *2-30*
2.10.3 Refresh *2-34*
2.10.4 Shading *2-34*
2.10.5 Drawing Grid Options *2-35*
2.10.6 Auto vs. Manual Redraw *2-35*
2.10.7 Hide/Show Hydrograph Window *2-36*
2.11 VIEW MENU 2-36
2.11.1 Frame Image *2-36*
2.11.2 Set Window Bounds *2-36*
2.11.3 Z Magnification *2-36*
2.11.4 Changing the Viewing Angles *2-36*

3 MAP MODULE 3-1

3.1 INTRODUCTION 3-1
3.2 FEATURE OBJECTS 3-2
3.2.1 Creating Watershed Models Directly *3-2*
3.2.2 Creating TINs or Finite Difference Grids *3-3*
3.2.3 Mapping Curve Numbers and Other Analysis Parameters *3-3*
3.2.4 Feature Object Types *3-3*
3.2.5 Feature Object Tools *3-7*
3.2.6 Coverage Type *3-9*
3.2.7 Display Options *3-25*
3.2.8 Menu Commands *3-26*
3.2.9 CASC2D *3-37*
3.3 DRAWING OBJECTS 3-39
3.3.1 Drawing Object Tools *3-39*
3.3.2 Display Attributes *3-40*
3.3.3 Drawing Depth *3-41*
3.3.4 Drawing Order *3-41*
3.4 IMAGES 3-42
3.4.1 Display Options *3-43*
3.4.2 Importing Images *3-43*
3.4.3 Registering An Image *3-44*
3.4.4 Resampling Images *3-46*
3.4.5 Fit Entire Image *3-46*
3.4.6 Deleting Images *3-46*
3.4.7 Exporting Images *3-47*
3.4.8 Capturing The Screen *3-47*
3.5 DXF FILES 3-47
3.5.1 Display Options *3-47*

3.5.2 Importing DXF Files *3-48*
3.5.3 Deleting DXF Objects *3-48*
3.5.4 Converting DXF Objects to Feature Objects *3-48*
3.5.5 Converting DXF Objects to TINs *3-49*

4 DEMS 4-1

4.1 INTRODUCTION 4-1
4.2 TOOL PALETTE 4-1
Select DEM Points *4-2*
Contour Labels *4-2*
4.3 DEM DISPLAY OPTIONS 4-2
4.3.1 Points *4-3*
4.3.2 Contours *4-3*
4.3.3 Flat DEM Cells *4-4*
4.3.4 NODATA Cells *4-4*
4.3.5 Inactive Cells *4-4*
4.3.6 Land Use *4-4*
4.3.7 Soil Type *4-4*
4.3.8 Display Step *4-4*
4.4 IMPORTING DEMS 4-5
4.5 DEM POINT STATUS 4-5
4.5.1 Set Active Region *4-5*
4.5.2 Activate Selected *4-5*
4.5.3 Activate All *4-5*
4.5.4 Inactivate Selected *4-6*
4.5.5 Inactivate All *4-6*
4.5.6 Delete Inactive *4-6*
4.6 SMOOTHING DEMS 4-6
4.6.1 Smoothing Options *4-6*
4.6.2 Restoring Elevations *4-8*
4.6.3 Removing Original Elevations *4-8*
4.6.4 Interpolating DEM Elevations to TIN *4-8*
4.6.5 Filling in Gaps Between DEMs *4-9*
4.6.6 Lat Lon Calculator *4-9*

5 DRAINAGE - DEMS 5-1

5.1 INTRODUCTION 5-1
5.1.1 Limitations and Memory Considerations *5-2*
5.2 TOOL PALETTE 5-2
Flow Path *5-2*
Select Points/Nodes *5-2*
Select Vertex *5-2*
Select Arc *5-2*
Create Points *5-3*
Create Arc Vertex *5-3*
Create Arc *5-3*
Select Polygon *5-3*
Select Branch *5-3*
Select Network *5-3*
5.2.11 Place Drainage Label *5-4*
5.3 DISPLAY OPTIONS 5-4
5.3.1 Watershed *5-5*
5.3.2 Stream *5-5*

5.3.3 Flow Directions 5-6
5.3.4 Flow Accumulations 5-6
5.3.5 Color Filled Drainage Basins 5-6
5.3.6 Display Basin Patterns 5-6
5.3.7 Fill basin boundary only 5-6
5.3.8 Basin Data Attributes 5-6
5.4 FLOW DIRECTIONS & ACCUMULATIONS 5-7
5.5 DRAINAGE ANALYSIS 5-9
5.5.1 DEM Streams->Feature Arcs 5-9
5.5.2 Defining Basins 5-10
5.5.3 Inactivate Null Basin 5-11
5.5.4 DEM Basins Boundaries to Polygons 5-11
5.5.5 Drawing Flow Patterns 5-11
5.5.6 Computing Basin Data 5-11
5.5.7 Merge Selected Basins 5-12
5.5.8 Polygon Basin ID's -> DEM 5-12
5.6 TOPAZ 5-12
5.6.1 DEM Processing 5-13
5.6.2 Watershed Outlet 5-13
5.6.3 Aggregation/Resampling 5-14
5.6.4 Smoothing 5-14
5.6.5 Analysis and Adjustment of Outlet Depressions 5-14
5.6.6 Output Options 5-14

6 TINS 6-1

6.1 INTRODUCTION 6-1
6.2 TOOL PALETTE 6-1
Select Vertices 6-1
Select Triangles 6-2
Select TIN 6-2
Select Vertex Strings 6-2
Create Vertices 6-2
Create Triangles 6-3
Swap Edges 6-3
Contour Labels 6-3
Select Basins 6-3
Place Basin Labels 6-3
Flow Path 6-4
6.3 DISPLAY OPTIONS 6-4
6.3.1 Vertices 6-5
6.3.2 Triangles 6-5
6.3.3 Contours 6-5
6.3.4 Boundaries 6-5
6.3.5 Circumcircles 6-6
6.3.6 Vertex/Triangle Numbers 6-6
6.3.7 Vectors 6-6
6.3.8 Elevations 6-6
6.3.9 Ridge Edges 6-6
6.3.10 Channel Edges 6-6
6.3.11 Flat Triangles 6-6
6.3.12 Flat Ridge Edges 6-7
6.3.13 Flat Channel Edges 6-7
6.3.14 Pits 6-7

6.3.15 Soil Columns *6-7*
6.3.16 Split Flow *6-7*
6.3.17 Soil Group *6-7*
6.3.18 Land Use *6-8*
6.3.19 Hydrologic Soil Group Legend *6-8*
6.3.20 Land Use Legend *6-8*
6.4 TIN ATTRIBUTES 6-8
6.5 VERTEX OPTIONS 6-8
6.5.1 Creating New Vertices *6-8*
6.5.2 Deleting Vertices *6-9*
6.5.3 Changing Vertex Positions and Z Values *6-9*
6.5.4 Vertex Options Dialog *6-10*
6.5.5 Lock / Unlock Vertices *6-11*
6.5.6 Scaling Vertices *6-12*
6.5.7 Remove Duplicates *6-12*
6.6 TRIANGULATION 6-12
6.6.1 Triangulation Options *6-13*
6.6.2 Triangulate *6-14*
6.6.3 Subdivide TIN *6-14*
6.6.4 Merging TINs *6-15*
6.7 BREAKLINES 6-16
6.7.1 Breakline Options *6-16*
6.8 BOUNDARY TRIANGLES 6-17
6.8.1 Selecting Boundary Triangles *6-17*
6.8.2 Length Ratio *6-18*
6.9 AUTOMATED TIN EDITING 6-18
6.9.1 Interpolating Flat Triangles *6-19*
6.9.2 Smoothing Triangles *6-19*
6.9.3 Smoothing Pits *6-20*
6.9.4 Filtering A TIN *6-20*
6.9.5 Edge Swapping *6-23*
6.10 TIN BOUNDARY -> POLYGON 6-23
6.11 TIN -> SCATTER POINTS 6-24
6.12 TIN->DEM 6-24

7 DRAINAGE - TINS 7-1

7.1 INTRODUCTION 7-1
7.2 TOOL PALETTE 7-2
7.3 DRAINAGE DISPLAY OPTIONS 7-2
7.3.1 Outlets *7-3*
7.3.2 Outlet Names *7-3*
7.3.3 Stream Networks *7-3*
7.3.4 Pipe Color *7-3*
7.3.5 Reservoirs *7-3*
7.3.6 Gages *7-3*
7.3.7 Thiessen Polygons *7-4*
7.3.8 Basin Centroids *7-4*
7.3.9 Drainage Basin Boundaries *7-4*
7.3.10 Fill Drainage Basins *7-4*
7.3.11 Basin IDs *7-4*
7.3.12 Basin Names *7-4*
7.3.13 Basin CN's *7-5*
7.3.14 Number of Triangles/Flowpath *7-5*

7.3.15 Watershed Color *7-5*
7.3.16 Downhill Overland Color *7-5*
7.3.17 Downhill Channel Color *7-5*
7.3.18 Uphill Overland Color *7-5*
7.3.19 Uphill Channel Color *7-5*
7.3.20 Data Text Color *7-5*
7.3.21 Geometric Attributes *7-6*
7.4 OUTLETS 7-6
7.4.1 Find Default Outlets *7-6*
7.4.2 Add Outlets *7-6*
7.4.3 Delete Outlets *7-6*
7.4.4 Select Branching Nodes *7-6*
7.5 STREAMS 7-7
7.5.1 Create Streams *7-7*
7.5.2 Create Pipe *7-8*
7.5.3 Delete Stream Segments *7-8*
7.5.4 Delete All Streams *7-8*
7.5.5 Display Stream Profile *7-8*
7.6 RESERVOIRS 7-8
7.6.1 Reservoirs and Storage Capacity Curves *7-9*
7.6.2 Deleting Reservoirs *7-10*
7.7 DRAINAGE 7-10
7.7.1 Define Basins *7-10*
7.7.2 Refine Boundaries *7-10*
7.7.3 Correct Split Flow Vertices *7-11*
7.7.4 Merge Basins *7-11*
7.7.5 Split Basins *7-11*
7.7.6 Delete All Basin Data *7-11*
7.7.7 Delete Null Basin Triangles *7-11*
7.7.8 Draw Flow Patterns *7-12*
7.7.9 Draw Watershed *7-12*
7.7.10 Compute Basin Data *7-12*
7.7.11 Computing Area Between Elevations *7-13*
7.8 CONVERTING DRAINAGE DATA TO FEATURE OBJECTS (GIS DATA) 7-13

8 FLOOD PLAIN ANALYSIS 8-1

8.1 INTRODUCTION 8-1
8.2 TOOL PALETTE 8-2
8.3 FLOOD PLAIN DISPLAY OPTIONS 8-2
8.3.1 Stage Height *8-3*
8.3.2 Fixed Stage *8-3*
8.3.3 Fixed Stage Values *8-3*
8.3.4 Interpolated Stage *8-3*
8.3.5 Interpolated Stage Values *8-3*
8.3.6 Flood Plain Boundary *8-3*
8.3.7 Flood Plain Color Filled *8-4*
8.3.8 Flood Plain Contours *8-4*
8.4 STAGE 8-4
8.4.1 Read Stage *8-4*
8.4.2 Edit Stage *8-5*
8.4.3 Remove Stage *8-5*
8.4.4 Interpolate Stage *8-5*
8.4.5 Adding Stage Streams *8-6*

8.4.6 Save Stage *8-6*
8.5 FLOOD PLAIN OPTIONS 8-7
8.5.1 Minimum Flood Plain Value *8-7*
8.5.2 Flood Plain Interval *8-8*
8.5.3 Number of Nearest Vertices with Stage *8-8*
8.5.4 Stage Import Options *8-8*
8.6 FLOODPLAINS 8-8
8.6.1 Delineate Flood *8-9*
8.6.2 Exporting to a GIS coverage *8-9*
8.6.3 Delete Flood Plain *8-10*

9 TOPOLOGICAL TREES 9-1

9.1 INTRODUCTION 9-1
9.2 TOOL PALETTE 9-2
Select Outlets *9-2*
Select Basins *9-2*
Select Diversion *9-2*
Select Hydrographs *9-2*
9.3 TREE DISPLAY OPTIONS 9-2
9.3.1 Frame Tree *9-3*
9.3.2 Collapsing the Topologic Tree *9-3*
9.4 DEFAULT MODEL 9-4
9.5 CREATING TREES 9-4
9.5.1 Add Outlet *9-4*
9.5.2 Insert Outlet *9-4*
9.5.3 Delete Outlet *9-5*
9.5.4 Add Basin *9-5*
9.5.5 Delete Basin *9-5*
9.5.6 Add Reservoir *9-5*
9.5.7 Delete Reservoir *9-5*
9.5.8 Add Diversion *9-5*
9.5.9 Retrieve Diversion *9-6*
9.5.10 Delete Diversion *9-6*
9.6 VIEWING HYDROGRAPHS 9-6
9.6.1 Display Options *9-7*
9.6.2 Reading Hydrographs *9-7*
9.6.3 Deleting Hydrograph Sets *9-8*
9.6.4 List Hydrographs *9-8*
9.7 WMS TREE FILE 9-9

10 HEC-1 INTERFACE 10-1

10.1 INTRODUCTION 10-1
10.1.1 Create a Topologic Tree *10-1*
10.1.2 Define Job Control Parameters *10-1*
10.1.3 Edit Basin/Outlet/Reservoir/Diversion Data *10-1*
10.1.4 Model Check *10-2*
10.1.5 Run HEC-1 ..... *10-2*
10.1.6 View Hydrographs *10-2*
10.2 TOOL PALETTE 10-2
10.3 JOB CONTROL 10-3
10.3.1 Name (ID) *10-3*
10.3.2 Day, Month, Year (IT) *10-4*
10.3.3 Computational Time Interval (IT) *10-4*

10.3.4 Beginning Time (IT) *10-4*
10.3.5 Number of Ordinates (IT) *10-4*
10.3.6 Output Control Options (IO) *10-4*
10.3.7 Units (IM) *10-4*
*10.3.8 Tree Diagram (*DIAGRAM)* *10-5*
*10.3.9 Listing (*LIST)* *10-5*
10.3.10 DEPTH/AREA STORMS (JD) *10-5*
10.3.11 MULTI-FLOOD STORMS (JR) *10-7*
10.3.12 Unit Graph and Loss Rate Optimization (OU) *10-8*
10.3.13 Routing Optimization (OR) *10-8*
10.4 ENTERING/EDITING HEC1 PARAMETERS 10-9
10.5 OUTPUT CONTROL(KO) 10-10
10.5.1 Routed and Combined Hydrographs at Outlets *10-10*
10.5.2 Comment Lines (KM) *10-10*
10.5.3 Output Control (KO) *10-11*
10.6 BASIN DATA 10-11
10.6.1 Basin Name (KK) *10-12*
10.6.2 Basin Area (BA) *10-12*
10.6.3 Direct Input Hydrograph (QI) *10-13*
10.6.4 Observed Hydrograph (QO) *10-13*
10.6.5 Base Flow (BF) *10-13*
10.7 PRECIPITATION 10-13
10.7.1 No precipitation *10-14*
10.7.2 Basin Average (PB) *10-14*
10.7.3 Gage (PG, PT, PW, PR) *10-14*
10.7.4 Hypothetical Storms (PH) *10-15*
10.8 LOSS METHODS 10-15
10.8.1 Uniform (LU) *10-16*
10.8.2 Exponential (LE) *10-16*
10.8.3 Green Ampt (LG) *10-16*
10.8.4 Holtan (LH) *10-17*
10.8.5 SCS (LS) *10-17*
10.9 UNIT HYDROGRAPH METHODS 10-18
10.9.1 Clark (UC) *10-18*
10.9.2 Snyder (US) *10-19*
10.9.3 SCS Dimensionless (UD) *10-20*
10.9.4 Given Unit Hydrograph (UI) *10-20*
10.9.5 Kinematic Wave (UK) *10-20*
10.10 SNOW MELT DATA 10-21
10.10.1 Elevation Zone Data (MA) *10-22*
10.10.2 Temperature Time Series (MT) *10-23*
10.10.3 Dew Point (MD), Shortwave Radiation (MS), and Wind Speed (MW) Series *10-23*
10.10.4 Losses (LM) *10-23*
10.11 ROUTING DATA 10-23
10.11.1 Outlet Names (KK) *10-24*
10.11.2 No Routing (RN) *10-24*
10.11.3 Muskingum (RM) *10-25*
10.11.4 Storage (RS) *10-25*
10.11.5 Straddler/Stagger (RT) *10-28*
10.11.6 Kinematic Wave (RK) and Muskingum-Cunge (RD) *10-29*
10.11.7 Channel Losses (RT) *10-29*
10.11.8 Direct Input Hydrographs (QI) *10-30*
10.11.9 Observed Hydrograph (QO) *10-30*

10.11.10 Pattern Hydrograph (QP) *10-30*
10.12 RESERVOIRS 10-30
10.13 DIVERSIONS 10-30
10.13.1 Editing Diversion Data (DT) *10-31*
10.14 GAGES (PG) 10-32
10.14.1 Defining/Editing Gages *10-32*
10.14.2 The Gage Position *10-33*
10.14.3 The Gage Type *10-34*
10.15 READING AND WRITING *HEC-1* FILES 10-34
10.16 RUNNING AN *HEC-1* ANALYSIS 10-34
10.16.1 Model Check *10-35*
10.16.2 Run HEC-1 ..... *10-35*
10.17 WMS AND HMS 10-36

11 TR-20 INTERFACE 11-1

11.1 INTRODUCTION 11-1
11.1.1 Create a Topologic Tree *11-1*
11.1.2 Define Job Control Parameters *11-1*
11.1.3 Edit Basin/Outlet/Reservoir/Diversion Data *11-1*
11.1.4 Model Check *11-2*
11.1.5 Run TR-20 ..... *11-2*
11.1.6 View Hydrographs *11-2*
11.2 TOOL PALETTE 11-2
11.3 JOB CONTROL 11-2
11.3.1 Title *11-3*
11.3.2 Main Time Increment *11-3*
11.3.3 Starting Time *11-3*
11.3.4 Unit Hydrograph Definition *11-3*
11.3.5 Precipitation *11-4*
11.3.6 Base Flow *11-4*
11.4 ENTERING/EDITING *TR-20* PARAMETERS 11-4
11.5 OUTPUT CONTROL 11-5
11.5.1 Peak Discharge and Runoff Volume *11-6*
11.5.2 Discharge Hydrograph *11-6*
11.5.3 Elevation of Hydrograph *11-6*
11.5.4 Flow Duration Data *11-6*
11.5.5 Save Results for Summary Tables *11-6*
11.6 BASIN DATA (RUNOFF) 11-6
11.6.1 Basin Name *11-7*
11.6.2 Basin Area *11-7*
11.6.3 SCS Curve Number *11-7*
11.6.4 Time of Concentration *11-8*
11.6.5 Basin Geometric Attributes *11-8*
11.6.6 Compute TC – Basin Data *11-8*
11.6.7 Compute TC – Map Data *11-8*
11.6.8 Use Defined Cross Section *11-8*
11.6.9 Use Defined Reservoir *11-9*
11.7 ROUTING DATA (REACH) 11-9
11.7.1 Outlet Names *11-10*
11.7.2 Reach Length *11-10*
11.7.3 Routing Method *11-10*
11.7.4 Output Control *11-12*
11.7.5 Direct Input Hydrographs *11-12*

11.8 RESERVOIRS 11-12
11.8.1 Name *11-14*
11.8.2 Defining Routing *11-14*
11.8.3 Start Routing Elevation *11-14*
11.8.4 Defining Reservoir Data *11-14*
11.8.5 Input Hydrograph *11-14*
11.8.6 Output Control *11-14*
11.9 DIVERSIONS 11-14
11.9.1 Name *11-15*
11.9.2 Diversion Inflow Parameters *11-15*
11.9.3 Diversion Outflow Parameters *11-15*
11.9.4 Output Control *11-16*
11.10 READING AND WRITING *TR-20* FILES 11-16
11.11 RUNNING A *TR-20* ANALYSIS 11-16
11.11.1 Model Check *11-17*
11.11.2 Run TR-20 ..... *11-17*

12 TR-55 INTERFACE 12-1

12.1 INTRODUCTION 12-1
12.2 INPUT REQUIREMENTS FOR *TR-55* 12-1
12.2.1 Initializing a TR-55 Simulation *12-2*
12.2.2 Basin Data *12-2*
12.2.3 Outlet Data *12-2*
12.3 USING THE *WMS* INTERFACE TO RUN *TR-55* SIMULATIONS 12-2
12.3.1 Basin Name *12-4*
12.3.2 Time of Concentration *12-4*
12.3.3 Drainage Area *12-4*
12.3.4 Curve Number *12-4*
12.3.5 Rainfall *12-5*
12.3.6 Pond/Swamp Factor *12-5*
12.3.7 Travel Time *12-5*
12.4 COMPUTING HYDROGRAPHS 12-5
12.5 COPYING RESULTS TO THE CLIPBOARD 12-6

13 RATIONAL METHOD 13-1

13.1 INTRODUCTION 13-1
13.2 RATIONAL METHOD EQUATION 13-2
13.2.1 Important Limitations *13-3*
13.3 COMPUTING PEAK FLOWS WITH THE RATIONAL METHOD EQUATION 13-4
13.3.1 Editing Basin Data *13-4*
13.3.2 Editing Outlet Data *13-5*
13.3.3 Determining Rainfall Intensities *13-6*
13.3.4 Computing Hydrographs with the Rational Method *13-9*
13.4 SAVING AND RESTORING A SIMULATION 13-16
13.5 MODRAT 13-17
13.6 2-YEAR 5-MINUTE PRECIPITATION (HYDRO-35) 13-18
13.7 2-YEAR, 15-MINUTE PRECIPITATION (HYDRO-35) 13-19
13.8 2-YEAR, 60-MINUTE PRECIPITATION (HYDRO-35) 13-20
13.9 100-YEAR, 5-MINUTE PRECIPITATION (HYDRO-35) 13-21
13.10 100-YEAR, 15-MINUTE PRECIPITATION (HYDRO-35) 13-22
13.11 100-YEAR, 60-MINUTE PRECIPITATION (HYDRO-35) 13-23

14 NATIONAL FLOOD FREQUENCY PROGRAM 14-1

14.1 INTRODUCTION 14-1
14.2 NFF RUN SIMULATION DIALOG 14-2
14.2.1 Basin Name *14-3*
14.2.2 State *14-3*
14.2.3 Total Watershed Area *14-3*
14.2.4 Maximum Flood Region *14-3*
14.2.5 Regional Regression Equations *14-3*
14.2.6 Using a Custom Defined Equation *14-5*
14.2.7 Computing Peak Discharges *14-5*
14.3 SAVING AND RESTORING A SIMULATION 14-6
14.4 MAXIMUM FLOOD REGION BOUNDARIES 14-6

15 HYDROLOGIC/HYDRAULIC CALCULATORS 15-1

15.1 INTRODUCTION 15-1
15.2 COMPUTING SCS CURVE NUMBERS, RUNOFF COEFFICIENTS, AND GREEN & AMPT INFILTRATION PARAMETERS 15-1
15.2.1 Computation Method *15-3*
15.2.2 Soil Type *15-4*
15.2.3 Land Use *15-4*
15.2.4 Computation Step *15-4*
15.2.5 Mapping Table *15-4*
15.3 COMPUTING TRAVEL TIMES (LAG AND TIME OF CONCENTRATION) 15-5
15.3.1 Computing Travel Times from Basin Data *15-6*
15.3.2 Computing Travel Times from Map Data *15-24*
15.4 CHANNELS 15-34
15.5 WEIRS 15-35
15.6 DETENTION BASINS 15-36
15.6.1 Hydrograph *15-38*
15.6.2 Storage-Capacity Curve *15-38*
15.6.3 Elevation-Discharge Relationship *15-39*
15.7 CULVERT ANALYSIS WITH HY8 15-40
15.7.1 HY8 General Specifications and Limitations *15-41*
15.7.2 Culvert Data *15-42*
15.7.3 HY8 Tailwater Data *15-44*
15.7.4 HY8 Roadway Data *15-45*
15.7.5 Saving the Input File *15-46*
15.7.6 Single Culvert Analysis *15-46*
15.7.7 Minimizing a Culvert Width for a Specified Head *15-47*
15.7.8 Multiple Culvert Analysis and Overtopping *15-48*

16 DATA SETS 16-1

16.1 INTRODUCTION 16-1
16.2 DATA BROWSER 16-2
16.2.1 File I/O *16-3*
16.2.2 Active Data Set *16-3*
16.2.3 Elevations *16-4*
16.2.4 Deleting Data Sets *16-4*
16.2.5 Data Set Info *16-4*
16.3 DATA CALCULATOR 16-4
16.4 CONTOURS 16-5
16.5 FRINGES 16-5
16.6 MAPPING ELEVATIONS 16-6
16.7 FILM LOOP ANIMATION 16-6

16.7.1 Saving Film Loops *16-7*
16.7.2 Film Loop Playback *16-7*
16.7.3 Film Loop Setup *16-7*
16.8 GAGES 16-9
16.8.1 The Gages Dialog *16-9*
16.8.2 The Gage Tools *16-11*
16.8.3 The Gage Plot Manager *16-12*

17 2D GRIDS 17-1

17.1 INTRODUCTION 17-1
17.2 GRID TYPES 17-1
17.3 TOOL PALETTE 17-2
Select Cell *17-2*
Select i *17-3*
Select j *17-3*
Contour Labels *17-3*
The Create Gages Tool *17-3*
The Select Gages Tool *17-4*
17.4 DISPLAY OPTIONS 17-4
17.5 GRID GENERATION 17-5
17.5.1 Create Grid *17-5*
17.6 ACTIVE / INACTIVE CELLS 17-6
17.6.1 Activate Polygon Region *17-7*
17.7 DATA TYPE CONVERSION 17-7
17.7.1 Grid -> Scatter Points *17-7*
17.7.2 Grid -> TIN *17-8*

18 2D SCATTER POINTS 18-1

18.1 INTRODUCTION 18-1
18.2 SCATTER POINT SETS 18-1
18.3 INPUTTING SCATTER POINT SETS 18-2
18.4 SAVING SCATTER POINT SETS 18-2
18.5 TOOL PALETTE 18-2
Select Scatter Point *18-2*
Select Scatter Point Set *18-2*
18.6 DISPLAY OPTIONS 18-3
18.7 INTERPOLATION OPTIONS 18-4
18.7.1 Linear Interpolation *18-5*
18.7.2 Inverse Distance Weighted Interpolation *18-5*
18.7.3 Clough - Tocher Interpolation *18-13*
18.7.4 Natural Neighbor Interpolation *18-14*
18.8 INTERPOLATION TO GRIDS 18-19
18.9 INTERPOLATING RAINFALL TO BASIN CENTROIDS 18-19
18.10 BOUNDING GRID 18-19

19 CASC2D INTERFACE 19-1

19.1 INTRODUCTION 19-1
19.2 TYPICAL STEPS TO SET UP A CASC2D MODEL 19-1
19.3 TOOL PALETTE 19-2
19.4 READING AND WRITING CASC2D PROJECT FILES 19-2
19.4.1 Saving a CASC2D Project *19-3*
19.4.2 Reading a CASC2D Project *19-4*
19.5 CREATING A GRID 19-4

19.5.1 Manually Creating a Grid *19-5*
19.5.2 Creating a Grid from Feature Objects *19-5*
19.5.3 Importing a GIS Based Grid *19-6*
19.6 JOB CONTROL PARAMETERS 19-6
19.6.1 Initializing CASC2D Data *19-7*
19.6.2 Computation Time *19-7*
19.6.3 Units *19-7*
19.6.4 Outlet Cell Information *19-7*
19.6.5 Channel Routing Parameters *19-7*
19.6.6 Output Control *19-8*
19.7 PRECIPITATION 19-8
19.7.1 Uniform Rainfall *19-9*
19.7.2 Gages *19-9*
19.8 EDITING CASC2D MAPS 19-10
19.8.1 Importing a Map *19-10*
19.8.2 Exporting a Map *19-10*
19.8.3 Assigning a Constant to the Map *19-11*
19.8.4 Creating a Data Set from a Map *19-11*
19.8.5 Creating a Map from a Data Set *19-11*
19.8.6 Reclassification *19-11*
19.9 OVERLAND FLOW PROPERTIES 19-12
19.10 INFILTRATION PROPERTIES 19-13
19.11 EVAPOTRANSPIRATION 19-14
19.12 EDITING MAP ATTRIBUTES 19-14
19.13 DEFINING CHANNELS 19-15
19.14 THE CASC2D MODEL CHECKER 19-15
19.15 RUNNING A CASC2D MODEL 19-15
19.16 POST-PROCESSING RESULTS 19-16

20 FILE FORMATS 20-1

20.1 INTRODUCTION 20-1
20.2 WMS SUPER FILES 20-2
20.3 TIN FILES 20-3
20.4 DEM FILES 20-5
20.5 IMAGE FILE 20-7
20.6 2D GRID FILES 20-8
20.7 2D SCATTER POINT FILES 20-10
20.8 ASCII DATA SET FILES 20-12
20.8.1 Using Vector and Scalar Data with Grids *20-15*
20.9 BINARY DATA SET FILES 20-16
20.10 GRASS GRID FILES (CASC2D MAPS) 20-20
20.11 ARC/INFO® ASCII GRID FILES 20-20
20.12 CASC2D SOILS TABLE FILE 20-21
20.13 GAGE FILES 20-22
20.14 DIGITIZED STREAM FILES 20-25
20.15 AUTOCAD STREAM FILE FORMAT 20-26
20.16 STAGE FILES 20-27
20.17 LAND USE FILES 20-28
20.18 SOIL TYPE RUNOFF COEFFICIENT FILES 20-30
20.19 HYDROGRAPH FILES 20-31
20.20 TABULAR DATA FILE 20-32
20.20.1 Tabular Data with Multiple Data Sets *20-32*
20.20.2 Tabular Data with Transient Data Set *20-32*

20.21 USER-DEFINED RATIONAL METHOD DIMENSIONLESS HYDROGRAPH FILES 20-33
20.22 CUSTOM NFF EQUATIONS FILE 20-34
20.22.1 Simplified explanation of creating a custom equation file: *20-39*
20.23 XY SERIES FILES 20-41

21 THE XY SERIES EDITOR 21-1

21.1 INTRODUCTION 21-1
21.2 XY SERIES LIST 21-2
21.3 XY EDIT FIELDS 21-3
21.3.1 Delete *21-3*
21.3.2 Interpolate *21-4*
21.3.3 Update *21-4*
21.3.4 Insert *21-4*
21.3.5 Compress *21-4*
21.4 XY SERIES PLOT 21-4
21.4.1 The Plot Tools *21-4*
21.4.2 The Plot Macros *21-5*
21.5 DEFINING A RAINFALL SERIES (PI/PC) 21-5
21.6 DEFINING A HYPOTHETICAL STORM SERIES (PH) 21-6
21.7 DEFINING A TIME AREA CURVE (UA) 21-6
21.8 DEFINING A STORAGE ROUTING CURVE (RS) 21-7
21.9 DEFINING A CROSS-SECTION (RX/RY) 21-7
21.10 DEFINING DIVERSION DATA (DI/DQ) 21-7
21.11 DEFINING A UNIT HYDROGRAPH (UI) 21-7
21.12 DEFINING A GIVEN HYDROGRAPH (QI) 21-8

CHAPTER 1

Introduction

1.1 Overview

WMS is a comprehensive environment for hydrologic analysis. It was developed by the Environmental Modeling Research Laboratory of Brigham Young University in cooperation with the U.S. Army Corps of Engineers Waterways Experiment Station.

1.1.1 Running WMS for the First Time

The same executable downloaded as a demonstration version can be enabled to full capabilities. When running *WMS* the first time (and each time until you register) the registration wizard is the first dialog you will see (Figure 1-1).

Welcome to WMS
WMS
Watershed
Modeling
System
Thank you for using WMS. This product has not yet been enabled.
If you have purchased a license, you may enable WMS at this time.
Otherwise you may use the program in demo mode, with printing
and saving disabled.
Demo Mode
Enable

Figure 1-1 Register Wizard Dialog.

By choosing the *Demo Mode* button you will be running the *WMS* demonstration which allows you full access to *WMS* capabilities without the ability to print or save. If you have licensed *WMS* and have a hardware lock or need to get the security string so that you can obtain a password, you should choose the *Enable* button (see section 2.8.14 for more information on registering *WMS*).

1.1.2 Digital Terrain Modeling

In addition to providing interfaces to hydrologic modeling programs like HEC-1 and TR-20, the digital terrain modeling functions of *WMS* can be used to create terrain models using Geographic Information Systems (GIS) data, Triangulated Irregular Networks (TINs), or gridded digital elevation models (DEMs). These data can then be used to delineate watersheds, streams and sub-basins.

Once boundaries have been created, geometric attributes such as area, slope and runoff distances can be computed automatically. A topological tree representation of the watershed is created, and all data necessary to define an *HEC-1* (or other) model can be entered by selecting basins and outlet points. Many display options are provided to aid in modeling and understanding the drainage characteristics of terrain surfaces.

WMS was designed as a comprehensive hydrologic modeling system. An interface to a spatially distributed model, CASC2D, is part of the current version of *WMS*. However, this model is still under review and is not available for public release. Several of the two-dimensional grid and interpolation commands can be used in support of hydrologic modeling, but the CASC2D model itself is not available.

1.2 Fundamental *WMS* Modeling Concepts

WMS has evolved to the point that there are many tools and different ways to accomplish the same thing. For this reason it is a good idea to have an understanding of the "big picture" on how *WMS* can best be used to develop and run hydrologic models. While this section outlines the fundamental concepts that are needed to efficiently use *WMS*, the tutorials provide an excellent way to cement your understanding of how it operates.

The distinguishing difference between *WMS* and other applications designed for setting up hydrologic models like *HEC-1* and *TR-20* is its unique ability to take advantage of digital terrain data for hydrologic model development. *WMS* uses three primary data sources for model development:

1. Geographic Information Systems (GIS) Vector Data
2. Digital Elevation Models (DEMs) or Gridded Elevation Sets
3. Triangulated Irregular Networks (TINs)

1.2.1 Guidelines for Using Feature Object (GIS) Vector Data

GIS vector data includes points, lines, and polygons that are used in *WMS* to represent basins, streams, and key points such as outlets or culverts. In *WMS* we refer to this GIS data as *Feature Objects*, and tools for using them are included in the *Map Module*. Feature object data can be used by itself to create a watershed models for hydrologic analysis or as a companion in the development of watershed models with TINs and DEMs.

Many times it is not practical to obtain digital elevation data and perform an automated watershed characterization prior to setting up a hydrologic model. Watershed and sub-basin boundaries may already be known and stored as part of a GIS or CAD database, or it may be straight-forward to trace an existing map to define streams and basins. With *WMS*, properly structured hydrologic models can be created automatically from points, lines, and polygons. Since these data are often already developed and stored in a GIS, importing from *ARC/INFO*® and *ArcView*® (section 2.8.6), or DXF files (section 3.5) is easily done.

An *ArcView*® extension has been developed to make the transition from GIS to *WMS* as simple as possible. The extension (*WMS-Hydro*) and accompanying documentation may be downloaded from the *WMS* website. Using this extension, many of the processes outlined in the steps that follow can be done inside of *ArcView*® prior to importing data in *WMS*.

The following are the basic steps to create watershed models from GIS data within *WMS*.

1. Obtain a Map or Already Developed GIS or CAD Data

The first step is to obtain a map that defines the streams and basins which will be modeled. If such a map already exists digitally as a CAD drawing or as part of a GIS database then it can be imported directly and the next step skipped.

2. Digitize the Map

The map can then be digitized using a tablet and standard digitizing software outside of *WMS* and then imported as a CAD or GIS file, or it can be created using "heads up," or "on-screen digitizing" inside of *WMS*. In order to do heads up digitizing you will need one of two things: 1) digital elevation data that can be contoured by *WMS*, or 2) a scanned tiff image that can be read into *WMS* and used as a background map.

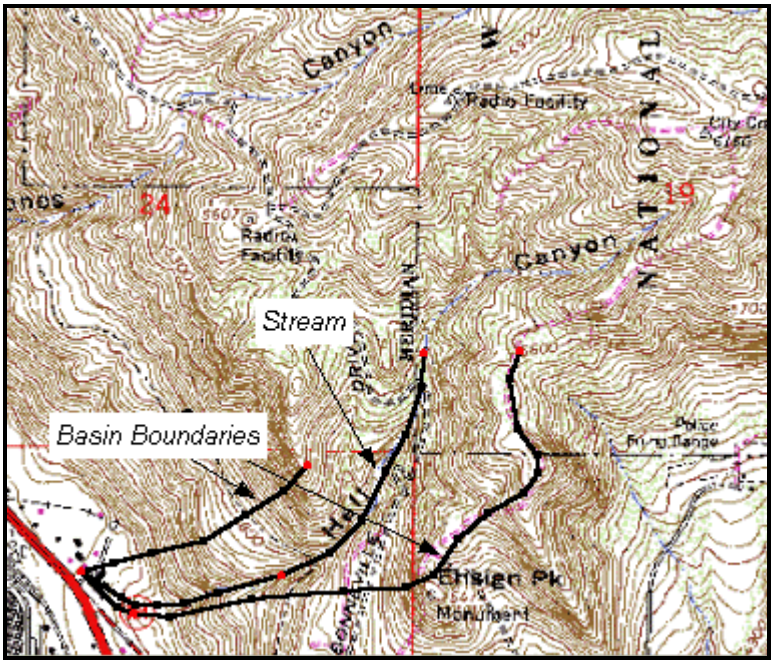


Figure 1-2 Imported TIFF Image Used as a Background Map for Digitizing.

3. Construct Feature Object Topology

The points and lines must be assigned the right attribute types, and the polygons for sub-basins constructed from the lines.

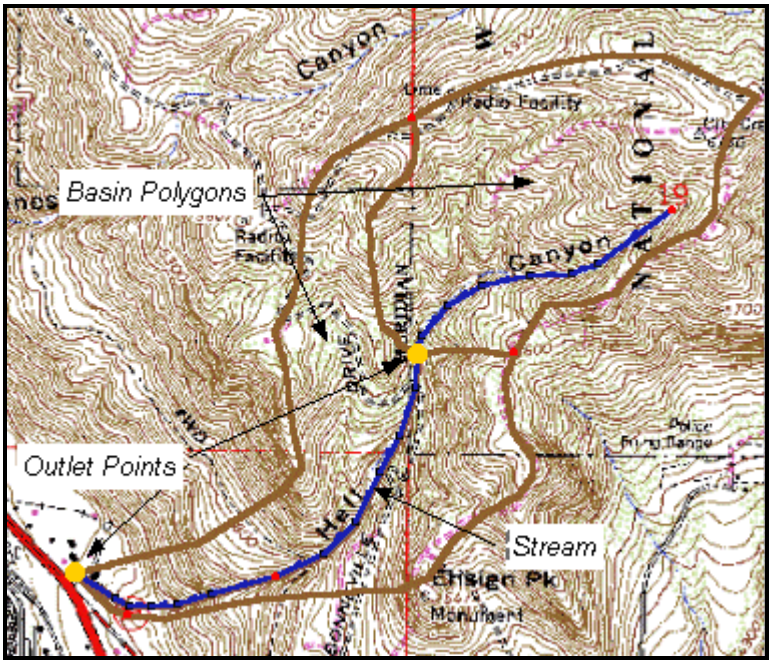


Figure 1-3 Feature Object Representation of a Watershed.

If you use data already developed in a GIS then you may have to do some editing. This will depend on how well the data being imported matches with the required data for watershed model development. In *WMS* three primary layers as illustrated in Figure 1-3: 1) A point layer representing the watershed outlet and any sub-basin outlet or confluence points, 2) a line layer representing a stream network, and 3) a polygon layer representing watershed boundaries.

If all three layers exist then construction of the watershed model topology can proceed, but if one or more of the layers are absent, they must be created manually from within *WMS*. For example if you only had a file that contained sub-basin boundaries, you would need to digitize the stream network and define the outlet locations of the sub-basins.

An important point to remember in *WMS* is that lines used to define a stream network have direction. For each line (arc) there is a beginning and an ending node and "flow" along the line is defined in this direction. When interactively creating lines in *WMS* you should always create streams from downstream to upstream. If you import a set of lines that has been previously created by another program you may discover that the order does not match what is required by *WMS*. The *Reorder Streams* (see page 3-28) command in the *Feature Objects* menu can be used to properly order the incoming lines so that they can be converted to streams.

4. Define the Hydrologic Model

Once the watershed model representation has been created, data defining a specific hydrologic model can be entered through a series of user friendly dialogs. Since *WMS* allows for all hydrologic modeling input to be defined separate from any digital terrain data, it is not required that the watershed model developed with feature objects be to scale. Area and length parameters can simply be manually defined using the model interface. Figure 1-4 shows how a topologic model is automatically created from point, line, and polygon feature object data.

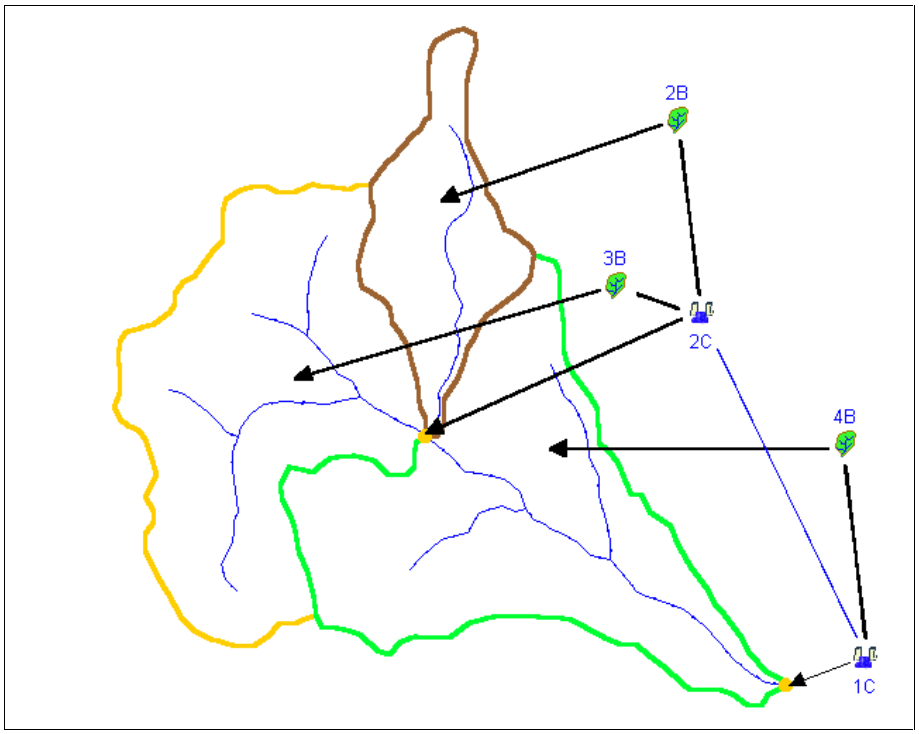


Figure 1-4 Topologic Model Automatically Constructed from Feature Objects.

1.2.2 Guidelines for Using DEM Data

The second method that can be used in *WMS* for defining watershed models and developing hydrologic data involves the use of digital elevation models or DEMs. A DEM (as defined in *WMS*) is simply a two-dimensional array of elevation points with a constant x and y spacing. While a DEM results in data redundancy for surface definition, its simple data structure and wide-spread availability have made them a popular source for digital terrain modeling and watershed characterization. Several researchers, including Puecker and Douglas (1975) and Garbrecht and Martz (1995) have developed methods to extract watershed geomorphology from DEMs. Beginning with version 5.0, *WMS* includes many of these same tools.

The two primary data sets that must be obtained to perform watershed delineation with DEMs are elevations, and flow directions. The most common form of DEM elevations are the USGS digital maps. DEMs can be downloaded free of charge from the EROS home page at http://edcwww.cr.usgs.gov/doc/edchome/ndcdb/ndcdb.html. Other sources of elevation data may include federal, state, and local government agencies, universities, or private data publishers. *WMS* can read digital elevation in standard USGS (the older single file format or the new SDTS formatted files), *ARC/INFO*®/*ArcView*® ASCII grid, and GRASS grid formats. Flow direction data for DEM points must be read in using the flow direction command in *ARC/INFO*®/*ArcView*®, GRASS, or by using the version of TOPAZ especially created for distribution with *WMS* (*Compute TOPAZ Flow Data…* command in the *Drainage* menu). All of these programs use a form of the eight point pour technique to determine the direction of flow. This technique specifies that the flow will be directed toward the neighboring (in a structured grid there are eight neighbors for each point) DEM point with the lowest elevation. The algorithms typically include functionality for eliminating pits and resolving ambiguities when the lowest elevation is shared by more than one neighboring point.

The typical steps for using DEMs to develop hydrologic models are:

1. Obtain and Import a Digital Elevation Model (DEM)

As mentioned above USGS DEMs can be downloaded from the internet, obtained from government agencies, universities, or private vendors. The *File Import* (section 2.8.6) command can be used to import the DEM from one of the supported formats. Figure 1-5 shows a contoured DEM after it has been imported.

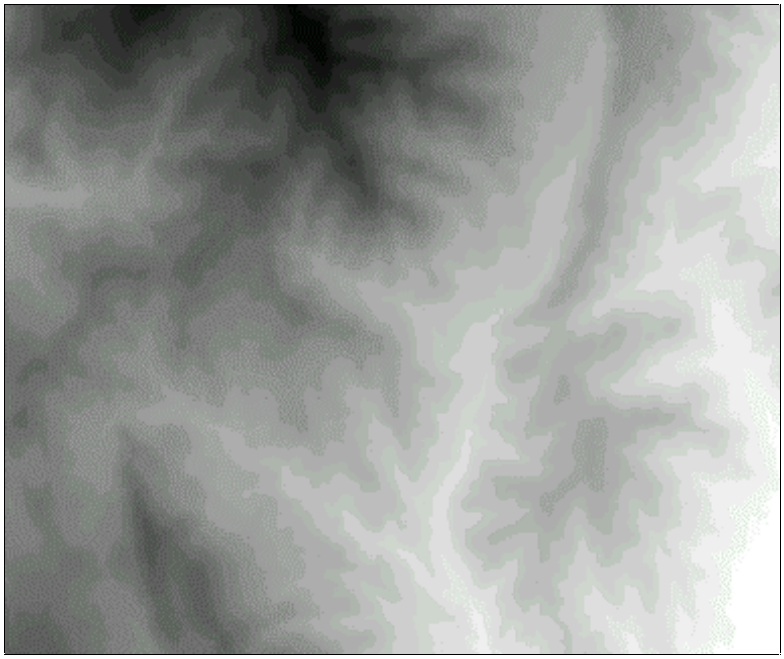


Figure 1-5 Contoured DEM.

2. Import a Flow Direction Grid

The flow direction grid can be computed from the active DEM region using a custom version of the TOPAZ model distributed with *WMS*. It can also be created in GRASS, *ARC/INFO*®, the *ArcView*® Spatial Analyst, or any other program that supports either *ARC/INFO*® ASCII or GRASS ASCII formats (section 5.3.3). The flow direction grid is then imported as a DEM point attribute, and is used to define the flow regime of the entire domain as illustrated in Figure 1-6.

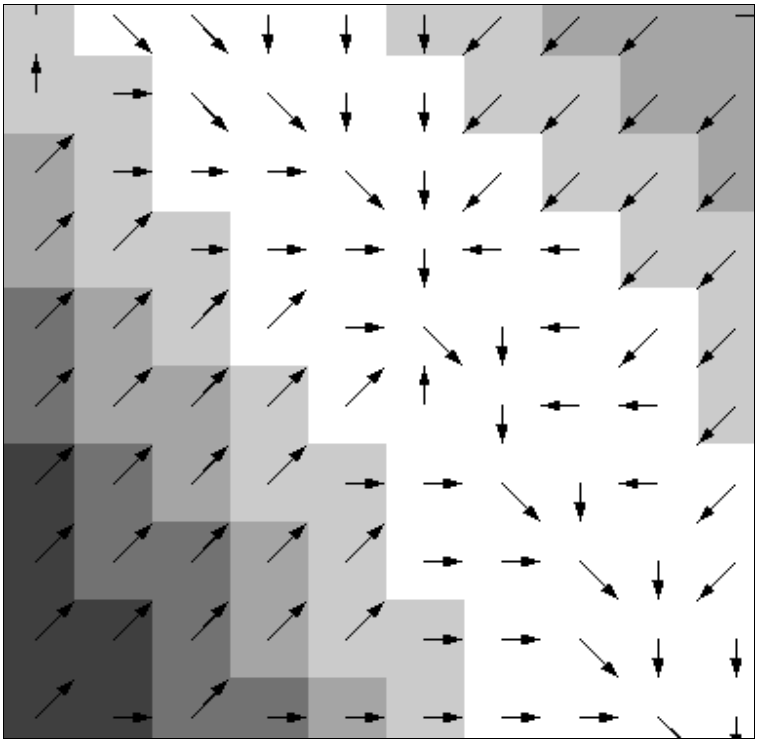


Figure 1-6 Flow Directions for a DEM.

4. Compute Flow Accumulations

With the flow directions assigned for each DEM point, the flow accumulation at each DEM point can be computed (section 5.3.4). The flow accumulation for a given DEM point is defined as the number of DEM points whose flow paths eventually pass through that point. For example, DEM points that are part of a stream have high flow accumulation values since the flow paths of all "upstream" points will pass through them. Streams are easily identified by displaying all DEM points with a flow accumulation value greater than a user-defined threshold as shown in Figure 1-7. Flow accumulations can be computed in WMS from the flow directions, or read from *ARC/INFO*®, GRASS, or TOPAZ formatted files.

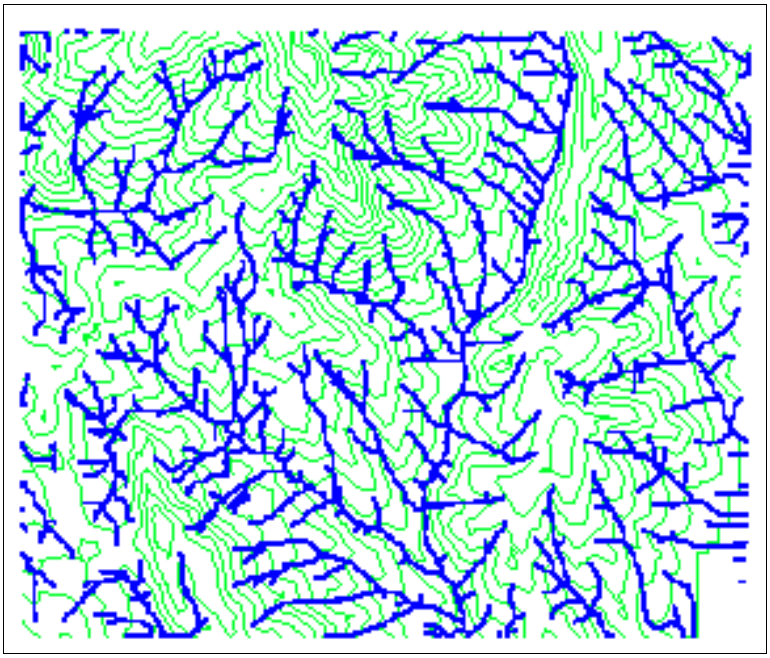


Figure 1-7 Flow Accumulation DEM Points Above a Defined Threshold.

5. Identify the Watershed Outlet and Convert DEM Streams to Arcs

With the aid of the flow accumulations, the location of the watershed outlet needs to be determined and an outlet feature point created there (section 5.2.5). A minimum threshold is then defined and all of the DEM points “upstream” from the defined outlet(s) are connected together to form a stream network of feature lines (section 5.5.1).

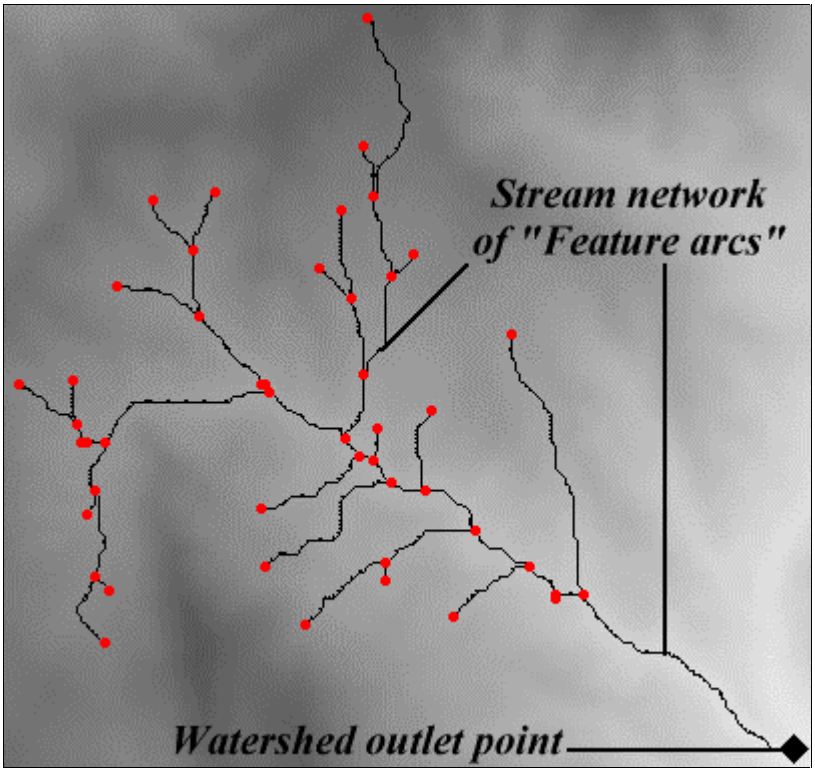


Figure 1-8 Stream Vector Created from Flow Accumulation Grid at a Specified Outlet Location.

You should note that the stream feature arcs can be created in any fashion. For example, in an urban area the streams will not likely be well-defined from the DEM elevations and flow directions. The flow directions for the DEM are then used for basic overland flow whereas the stream vectors are used for conveyance channels. Practically, you can think of *WMS* modifying the flow directions of the DEM points underlying the stream vectors so that flow always follows the defined stream vectors.

6. Define Interior Sub-basin Outlet Points

If you wish to further subdivide the watershed into sub-basins then nodes along the stream feature arcs should be converted to “outlet” nodes by using the feature point/node attributes dialog (see page 3-25). As these nodes are converted the hydrologic modeling tree is automatically updated.

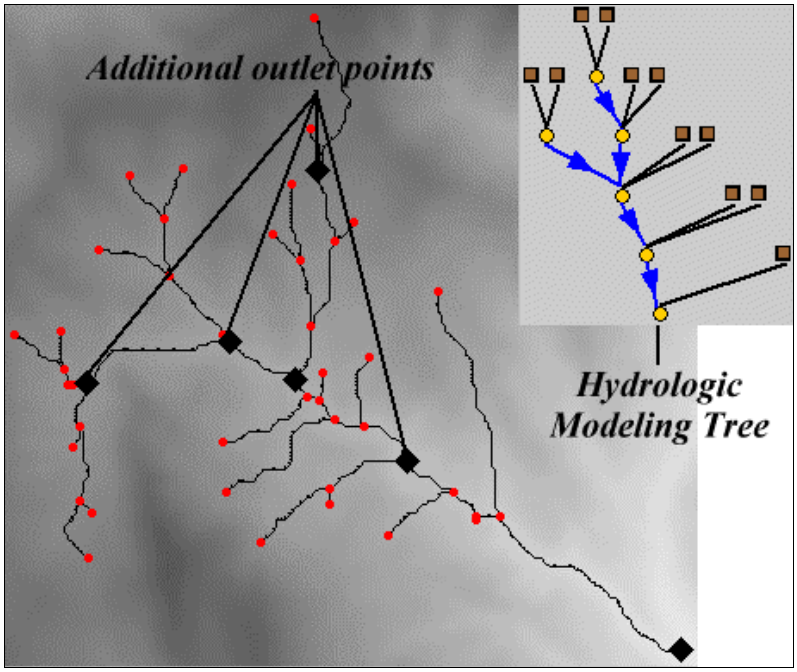


Figure 1-9 Interior Outlet Points Identified Along the Stream Arcs.

7. Define Basins

Using the outlets on the stream network and the flow directions, the contributing DEM points for each outlet are assigned the proper basin id (section 5.5.2).

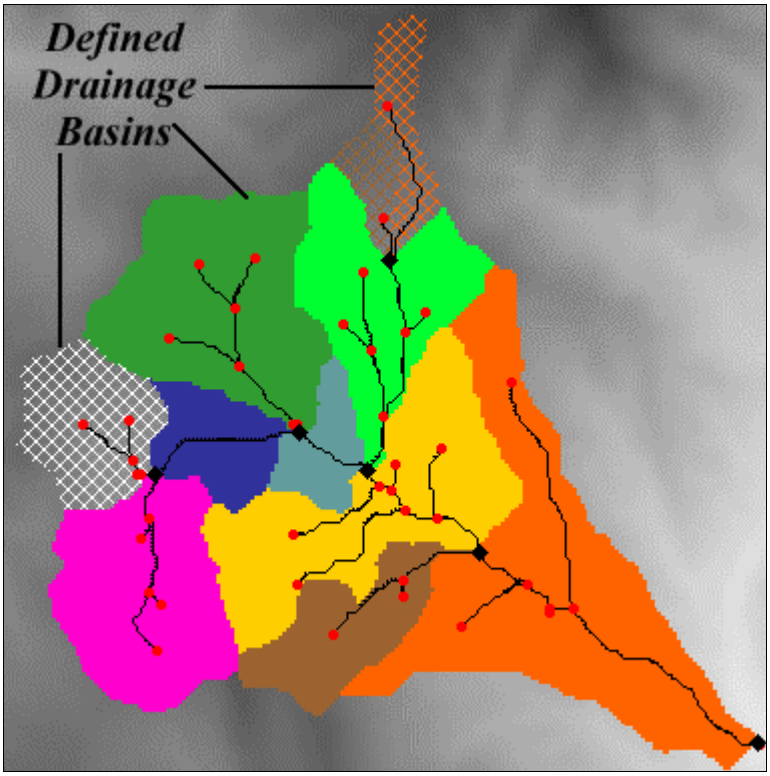


Figure 1-10 Basins Defined Using the DEM Flow Direction Grid and Stream Vectors.

8. Convert DEM Basins to Polygons

Similar to how flow accumulations were converted to stream arcs, the boundaries between DEM points with different basin ids can be converted to feature polygons (section 5.5.3). Storing a basin as a single polygon rather than several hundreds (or thousands) of DEM cells is much more efficient.

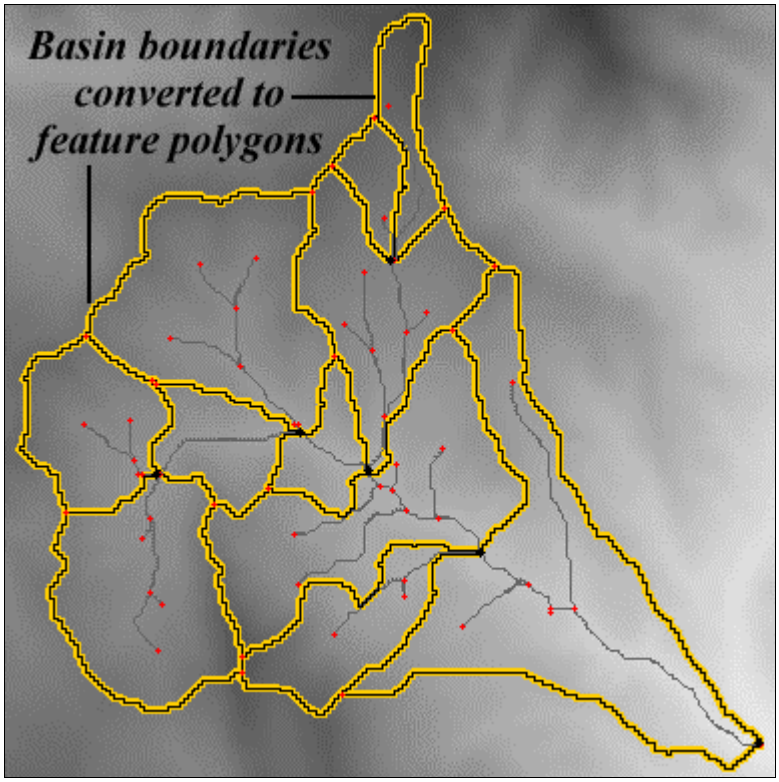


Figure 1-11 Gridded DEM Basins Converted to Feature Polygons.

9. Compute Basin Geometric Data

Once the boundaries of the sub-basins have been determined geometric properties important to hydrologic modeling (area, slopes, runoff distances, etc.) can be computed from the DEM data (section 5.5.6).

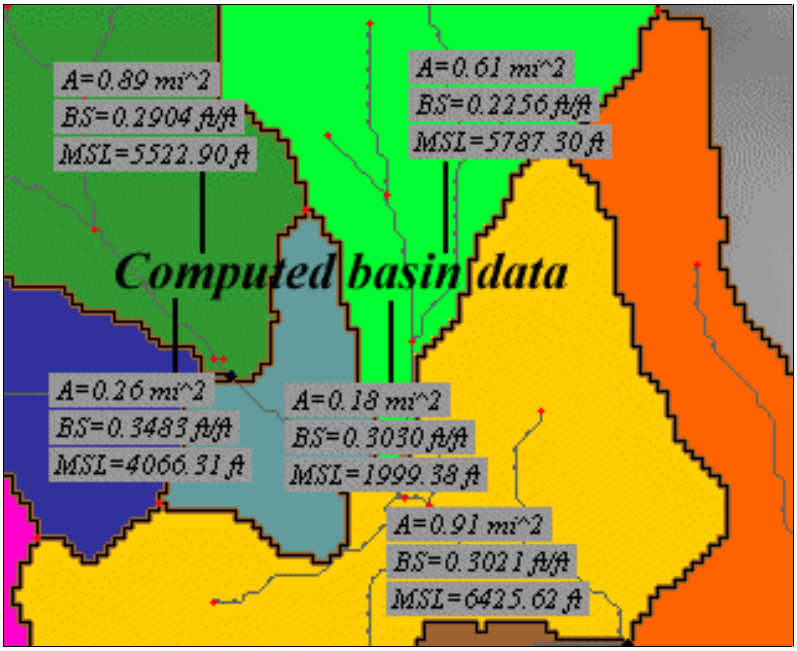


Figure 1-12 Some of the Computed Basin Attributes.

10. Define the Hydrologic Model

At this point you will have the same model as described in the previous section, where watersheds are defined strictly from the feature points, lines (arcs), and polygons. The computed data from step nine is automatically stored in the appropriate locations for hydrologic model definition, and the remaining parameters for the desired hydrologic model can be entered using the appropriate interface dialogs.

1.2.3 Guidelines for Using TIN Data

The third method, which has been the traditional approach in *WMS* for defining watershed models, is through the use of TINs. Developing watersheds from TINs often involves the use of both feature objects and DEMs. The following steps can be used as a guideline for watershed characterization with TINs.

1. Obtain Background Elevation

An elevation source is required for creating a TIN. If you have an existing TIN whose triangle edges already conform to key drainage features such as streams, then you can use the TIN and skip directly to step six (however, you must still define outlets and streams manually using the techniques described in section 7.5.1). Even if you already have a TIN data source we recommend that in most cases you use it as a background elevation source and a create a new TIN from it. A background elevation source can be a DEM, TIN, or both.

2. Smooth the Background Elevation

Digital elevation data is often rounded to the nearest integral value (foot or meter) for storage efficiency. However, this can cause problems for automated basin delineation techniques, especially where there is relatively little relief. *WMS* has utilities for smoothing both DEM (section 4.6) and TIN (section 6.9.4) background elevation data. The results can be dramatic as seen in Figure 1-13, and often make the difference in being able to successfully complete a watershed modeling project.

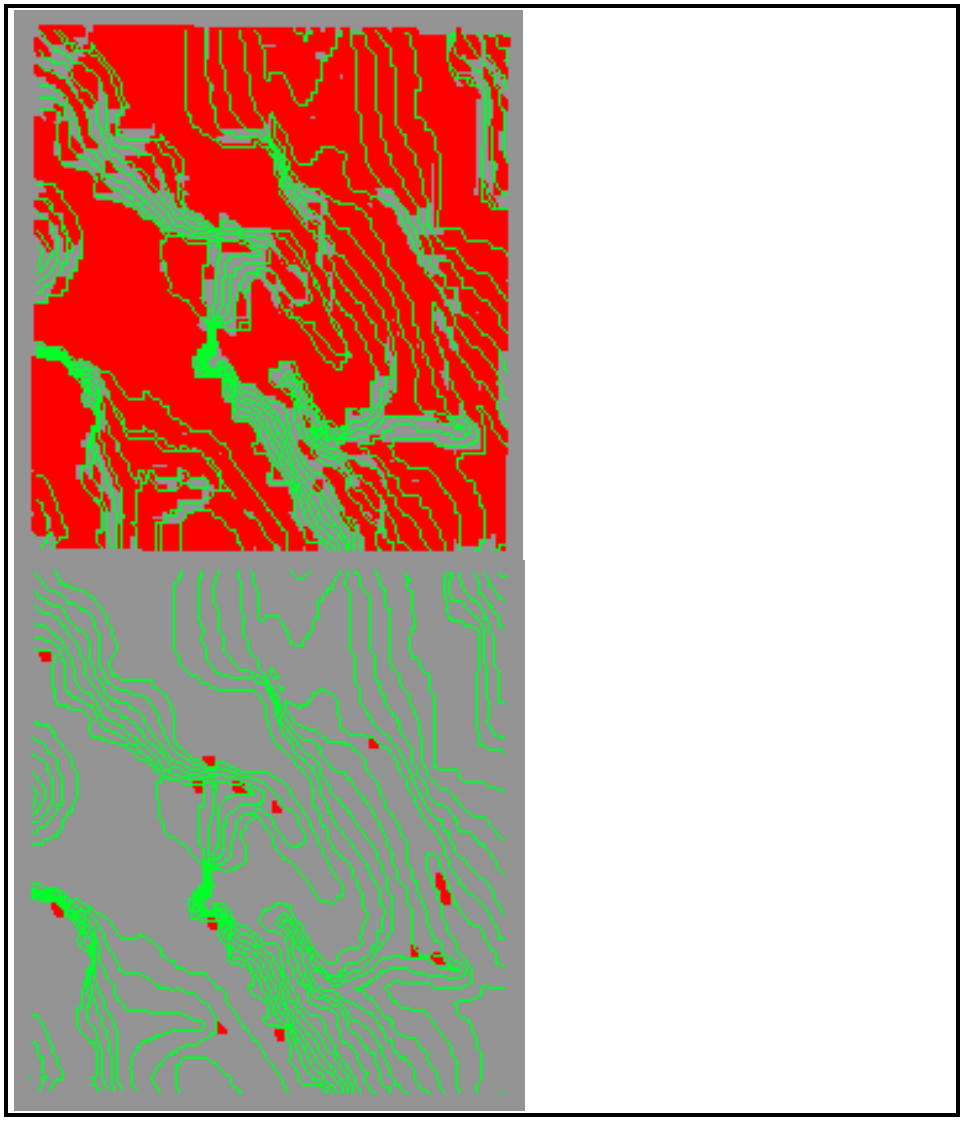


Figure 1-13 Flat DEM Data Before and After Smoothing.

3. Create a Conceptual Model with Feature Objects

In order to insure that triangle edges in the resulting TIN will conform to streams and other important drainage features, you need to identify them

with feature objects. A rough basin boundary defining the domain of the TIN region needs to be created. Additionally, any lines such as streams and roads that should be represented with triangle edges should also be created as part of the conceptual model.

Conceptual models can be created in many different ways, but some of the easiest ways in *WMS* include:

1. Import existing digital data in GIS, CAD (DXF), DLG, or other simplified xy formats (see section 2.8.6 on importing data).

2. Use a contour display of a DEM and on-screen digitize the rough boundary and stream arcs.

3. Use a registered image as a background for on-screen digitizing (section 3.4.3).

4. Use a DEM to define flow accumulations and then convert to stream arcs as described in step five of the previous section.

5. Use a hillshaded image as a background for on-screen digitizing (section 2.10.4). The *Screen Capture* command in the *Images* menu can be useful for saving a hillshaded DEM as an image file.

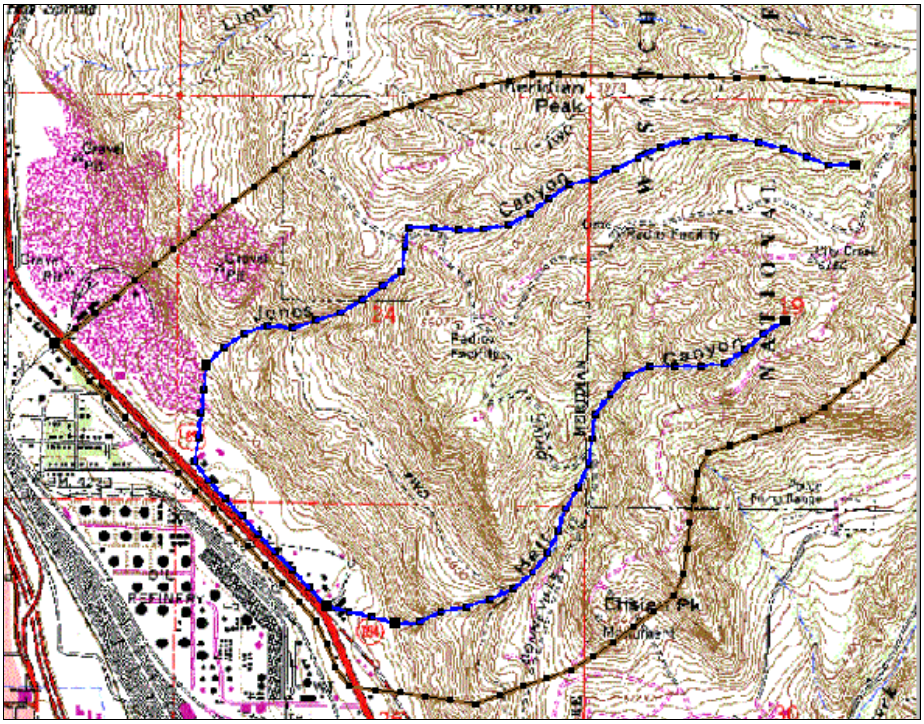


Figure 1-14 Conceptual Model Defined from Registered Image.

4. Redistribute Vertices

The density of vertices in the TIN created from the conceptual model and background elevation data can be controlled by the spacing of arc vertices in the conceptual model. *WMS* has tools to automatically increase or decrease arc vertex density (see page 3-28). The density may vary along the arcs, allowing for higher definition in some regions and lower in others.

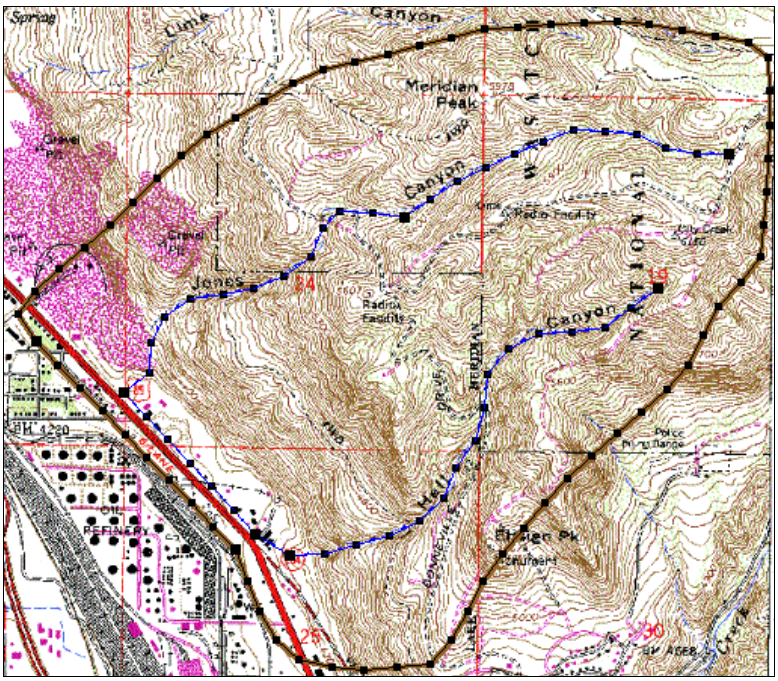


Figure 1-15 Redistribution of Arc Vertices.

5. Create TIN

TIN creation builds on the previous four steps. The outer polygon is used to define the limits or extents of the TIN. TIN vertices are created inside this polygon at a density proportional to the spacing of vertices on the nearest arc. After the TIN is created, the stream (and other interior) arcs are forced as breaklines so that they are honored in the TIN as triangle edges, and the elevations for the vertices are interpolated from the background elevation map (see page 3-33).

6. Edit TIN

Even though the newly created TIN conforms to the topographic features defined by feature arcs, there are inevitably some anomalies that must be corrected in order to use the TIN for basin delineation. These include flat triangles, flat edges, and pits. *WMS* contains several tools for both automatic (section 6.9) and manual (user interaction) (sections 6.2.7, □) elimination of these anomalies.

7. Complete Stream Network and Outlet Definitions

By default there may only be a single outlet point for the watershed defined, or perhaps only a portion of the stream network. *WMS* can be used to add additional outlet points (representing sub-basin outlets, culverts, etc.) (section 7.4.2) and stream branches (section 7.5.1) after the TIN has been created from the feature objects (see Figure 1-16). Even after delineation you can return to this step and redefine the locations where you would like sub-basins created.

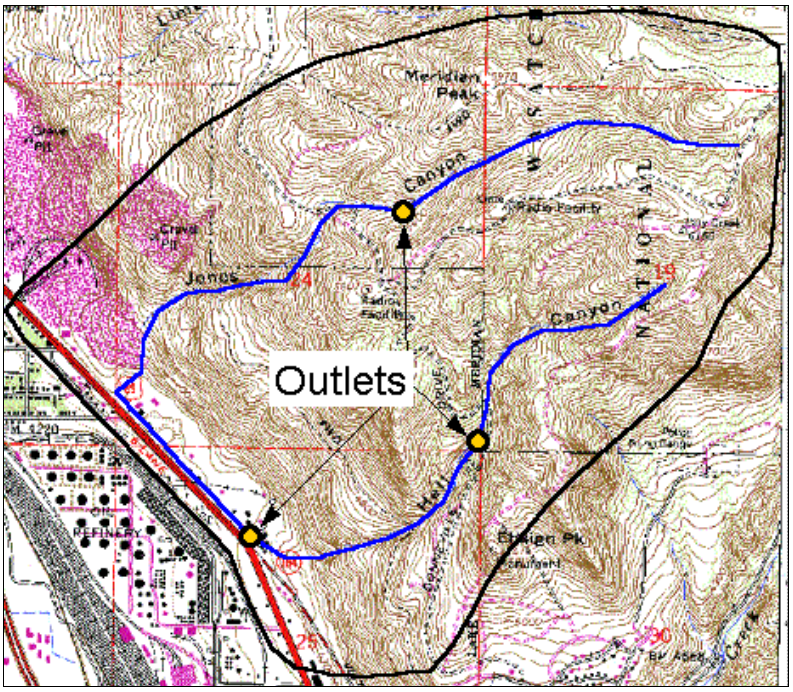


Figure 1-16 Addition of Outlet Points on the TIN to Define Sub-basins.

8. Delineate Basins

A flow path is initiated from the centroid of each triangle and followed until the first outlet point (section 7.7.1). The triangle is then assigned the basin ID corresponding to the branch of the stream the flow path entered from (by default, a separate basin is created for each upstream branch of an outlet as illustrated in Figure 1-17 where there are two basins for the outlet in the lower left corner). Basins can be merged (section 7.7.4) later, providing they both belong to the same outlet point.

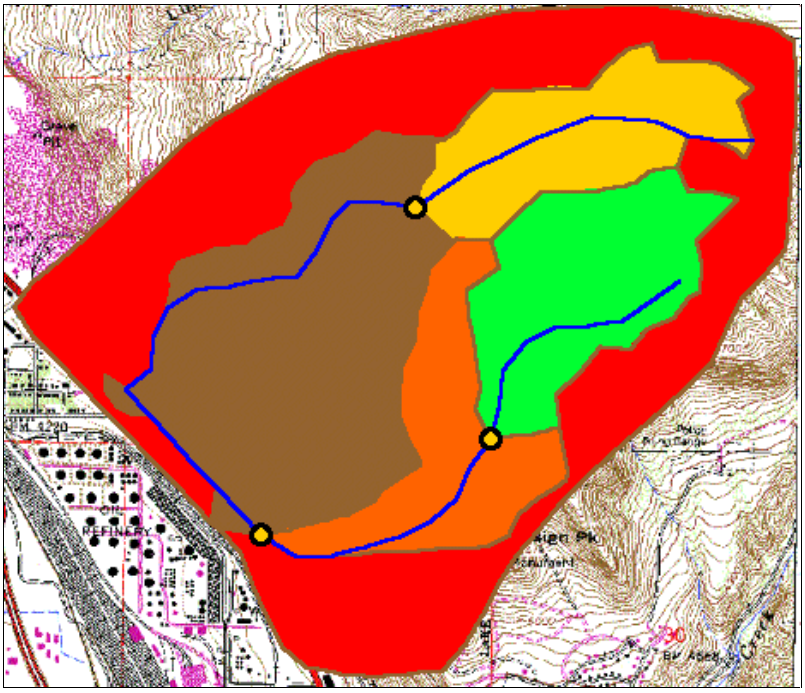


Figure 1-17 Delineated Sub-basins from a TIN.

9. Refine TIN

During the basin delineation process in step eight some problems with divergent or splitting flow paths may occur. Again, *WMS* has tools which will allow you to correct these problems automatically (section 7.7.3) and manually. The automatic method will correct the problem about 90% of the time, while in 10% of the cases you may need to swap edges or edit the TIN in some other way in order to eliminate the split flow vertices.

If there are no split flow vertices, or after you have corrected them, you will want to refine your boundaries and then eliminate triangles exterior to the actual watershed. Triangle basins are defined based on flow paths from the centroid of the triangle, so some triangles will actually straddle the basin boundary. The *Refine Boundaries* (section 7.7.2) command will split these triangles along the true boundary and results in "smoother" basin boundaries. Once you are satisfied with your watershed boundary the *Delete Null Basin Triangles* (section 7.7.7) command is used to eliminate all triangles whose flow paths do not pass through an outlet (basin).

10. Compute Basin and Stream Parameters

With the stream network and basin boundaries defined you can compute several important geometric parameters (section 7.7.10). These parameters (area, slope, length, etc.) are automatically tied to the hydrologic models (*HEC-1*, *TR-20*, etc.) where appropriate.

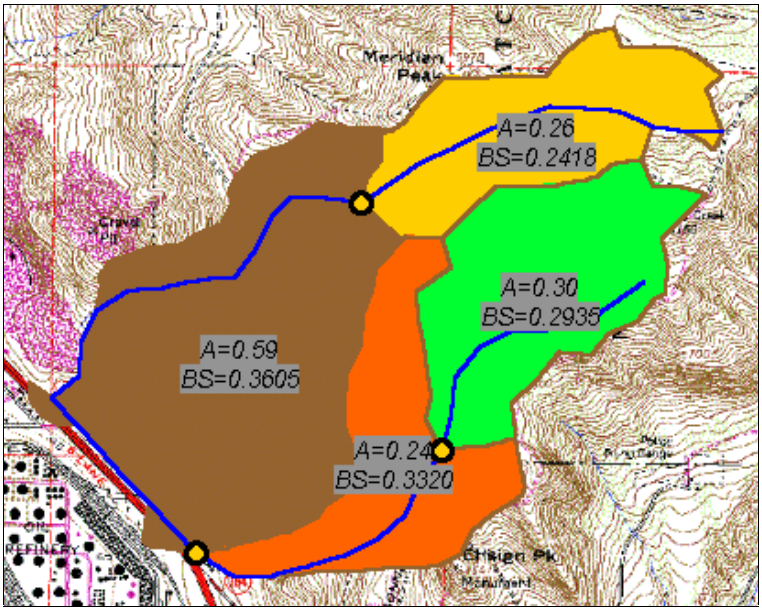


Figure 1-18 Computed Basin Geometric Parameters.

11. Define the Hydrologic Model

Along with the watershed definition on the TIN, an accompanying topologic model is created. You can then interact with the TIN or tree representation of the watershed to complete input for and run of the supported hydrologic models.

1.2.4 Guidelines for Using *ARC/INFO*® and *ArcView*® Formatted Data

Because the data structures used for the three primary methods of hydrologic data development in *WMS* parallel data types found in GIS software such as *ARC/INFO*® and *ArcView*®, there are several possible scenarios for importing and using data developed by GIS applications that support the *ArcView*® file formats.

Shape Files

There are just about as many different ways to have stored watershed data in a GIS as there are watersheds stored. Therefore, what you do with basin polygons or stream networks in *WMS* will likely be somewhat different. This section outlines some of the key issues involved in importing shapefiles (*ARC/INFO*® or *ArcView*® data) and provides examples of common problems.

Fundamental to importing any vector data layer is the ability to map attributes associated with the shape file to corresponding parameters used by *WMS*. The same dialog (Figure 1-19) is used for each of the three basic layer types. A set

of key words (shown in the table on page 2-10) can be used to define the item names of attributes in the shape file so that mapping to corresponding variables in *WMS* occurs automatically. In the event that the attribute name is different, the fields can be manually mapped. Regardless of the way your data is stored in the GIS, you should be able to take advantage of as much of the pre-defined and stored hydrologically related parameters as possible.

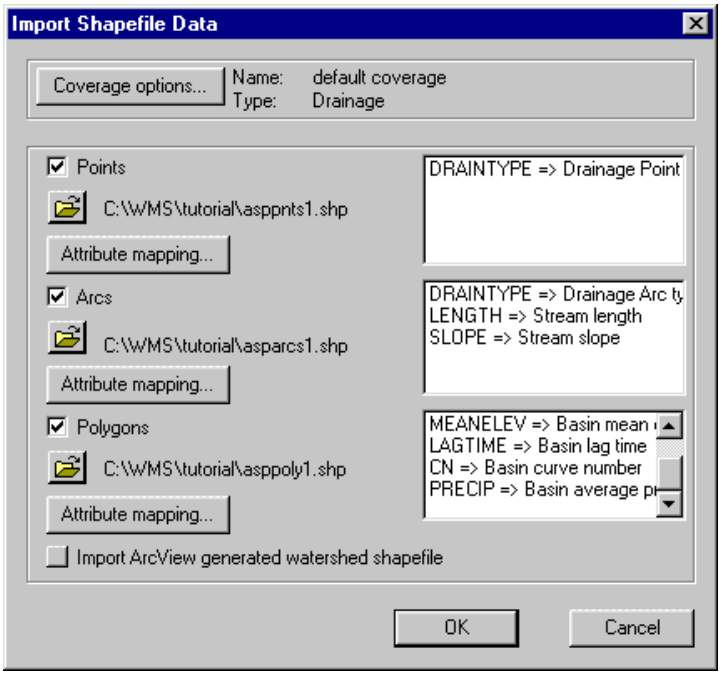


Figure 1-19 Import Shapefile Data Dialog.

Ideally you will have three data layers when importing watershed related shapefiles: 1) a polygon layer representing basin boundaries, 2) a line (arc) layer representing the stream network, and 3) a point layer representing the outlet points (these should be the intersection points of the basin polygons and stream network layers). If you have these three layers properly defined you should be able to import them and automatically create the topologic model used for hydrologic modeling in *WMS*. If one or more of these layers are not present you will need to either create it in *ArcView*® or *ARC/INFO*® or define it in *WMS* after importing what is available. For example if you only have a layer defining basin boundaries then you will need to construct a network using feature objects in *WMS* which properly "connects" the basin polygons together. This stream network may or may not actually represent the conveyance channel in the actual watershed, but must be present at least to the point that connectivity between sub-basins is defined.

Another problem that may occur is the ordering of vertices/nodes in a stream network may not be consistent with what *WMS* expects. *WMS* expects that the first or "from" node in an arc be the downstream node, while the second or "to" node is the most upstream node. If your stream vector data is not defined

in this fashion it will have to be reordered prior to generating a correct topological watershed representation. The *Reorder Streams* command can be used to accomplish this (see page 3-28). The tutorial in chapter four of the *WMS* tutorials demonstrates how to import/edit data coming from shapefiles.

A special extension for *ArcView*® has been developed by EMRL to allow some of these editing procedures to be taken care of on the GIS side. This extension also allows you to easily "package" your data in a superfile and then start *WMS* and pass the necessary data directly. See the *WMS* home page for more information about downloading the *ArcView*®extension with accompanying documentation.

Grid (DEM) Files

All *ARC/INFO*® or *ArcView*® grid data imported to *WMS* must be in the ASCII grid format. Grid files can be used as DEM's in *WMS*. Also, flow direction and flow accumulation grids can be used to define their respective attributes for DEM points. Once you have imported the elevation and flow direction ASCII grids, all of the remaining watershed parameters can be developed directly within *WMS*. The elevation DEM may also be used as the background elevation map used when creating a TIN.

TIN Files

ARC/INFO® TIN files must be in ASCII-NET format in order to import them and use for watershed characterization. The TIN can be used directly to create streams and basin boundaries, or as a background elevation map used in conjunction with feature objects for TIN creation.

1.3 Modules

The interface for *WMS* is divided into six separate modules. A module is provided for each of the basic processes supported by *WMS*. As you switch from one module to another, the *Tool Palette* and the menus change. This allows you to focus only on the tools and commands related to the process you are currently working on. Switching from one module to another can be done instantaneously to facilitate the simultaneous use of several processes when necessary. The modules listed in the next few paragraphs are supported in *WMS*.

1.3.1 Triangulated Irregular Network (TIN) Module

The *TIN* module is used for terrain modeling and automated basin delineation. TINs are formed by connecting a set of xyz points (scattered or gridded) with edges to form a network of triangles. Points used to create TINs can be obtained by digitizing a contour map (or a scanned image inside of *WMS*), or

generated automatically from feature arcs and polygons, using DEMs or existing TINs as background elevation maps. TINs can be contoured, displayed in oblique view with mapped images and hidden surfaces removed, and several other display options that can be set to visualize and understand the terrain surface better. TINs are used for basin delineation and drainage analysis. Basin areas and several other geometric parameters can be computed and combined with hydrologic analyses.

1.3.2 DEM Module

The *DEM* module is used to import and display USGS 7.5 minute (both the older single file format and the newer SDTS format) and 3 arc-second digital elevation models or to import a pre-processed DEM from either the *GRASS* or *ARC/INFO*® GIS. DEMs can be used to delineate watershed and sub-basin boundaries and then converted to a series of arcs and polygons. They can be contoured and displayed in oblique view. When creating a TIN, they can be used as a background elevation map.

1.3.3 Map Module

The *Map* module is used to define stream channels, ridges, boundaries, and any other important terrain features present in the model. The points, nodes, arcs, and polygons have been structured after the *ARC/INFO*® data model. TINs can be constructed from these feature objects using an existing TIN or a DEM as a background elevation map. Triangle edges are enforced along all arcs.

Land Use and Soil type layers can be created using feature objects in the *Map* module and then used to compute curve numbers or map other important modeling parameters. Other layers are also used for computing time of concentration or lag time, mapping rainfall and other parameters for the LA County F0601 model, and streams for 2-D analysis using *CASC2D*.

Within the *Map* module there are several other tools which can be helpful in either setting up models or presentation of results to a client. Tools for reading and writing of DXF files, mapping tiff images, and text annotation are part of this module.

1.3.4 Hydrologic Modeling (Tree) Module

Hydrologic analysis is typically done using lumped parameter models such as *HEC-1*. The *Tree* module provides a graphical interface to *HEC-1*, *TR-20*, *HSPF*, *TR-55*, Rational Method, the National Flood Frequency (NFF), and other programs. In the absence of terrain data, topological or tree representations of a watershed can be created. Then all necessary input data to run one of the supported models can be defined using a series of user-friendly

dialogs. This module is used for interfacing to hydrologic models and for the construction of topologic watershed models in the absence of digital terrain data.

1.3.5 Grid Module

The *Grid* module is used for surface visualization and for the development of a *CASC2D* rainfall/runoff analytical model. For example, the user can discretize a watershed into a number of grid cells and then define important rainfall, infiltration, and channel properties at grid cells in preparation for running *CASC2D*. Any parameter such as hydraulic conductivity or rainfall intensity may be interpolated from a set of scattered data points to the grid. Results of the 2D analysis can then be contoured on the grid or displayed with hidden surface removal and color fringes to display the variation in the computed results.

1.3.6 Scatter Point Module

The *Scatter Point* module is used to interpolate from groups of scattered data points to grids. The *Scatter Point* module can be used to interpolate from a set of scattered xy points representing something like rain gages to a finite difference grid or to basin centroids for establishing rainfall curves for *HEC-1*. A variety of interpolation schemes are supported.

1.4 Considerations For Reading Existing *HEC-1* Files

WMS is capable of reading *HEC-1* files that have been manually created using a text editor or some other program. However, there are a couple of problems which need to be considered, and may have to be altered either before or after reading in one of these files.

- There can be no blank fields in a file read in by *WMS*. If a field is left blank, *HEC-1* assumes the value of this field is 0. However, errors will occur or data will be lost if a file with a blank field is read into *WMS*.

- Names for all KK (Hydrograph Station identifying card) records must be unique. This problem won't surface until you run *HEC-1* with a new file created by *WMS* and try to read the hydrographs. In such cases all hydrographs will displayed at the first hydrograph station (KK record) with the duplicate name. This can be changed either inside or outside of *WMS*.

- *WMS* reads hydrograph results from the TAPE22 file. Many existing *HEC-1* files will not specify output to this file and you may need to define it for all hydrograph stations before you will be able to read in

the modeling results. This can be done by selecting all basins/outlets and bringing up the respective output control dialog.

- Only the first three ID cards are read into *WMS*.

- All KM cards (comments) are placed directly after the KK record for a hydrograph station. *WMS* will read KM cards from any position within the *HEC-1* file, but always writes them out directly following the KK card.

- *WMS* writes out a few other comment cards (preceded by an *) that are ignored by *HEC-1*. These cards are not necessary, but you should be aware of differences you might see from your original file and the one created by *WMS*.

1.5 Considerations For Reading Existing *TR-20* Files

WMS is capable of reading *TR-20* files created manually using a text editor or other program. However, there are a couple of problems which need to be considered, and may have to be altered either before or after reading in one of these files.

1. *WMS* reads analysis hydrograph results from the ".THY" file. Many existing *TR-20* files will not specify output to this file and you may need to define it for all hydrograph stations before you will be able to read in analysis results. This can be done by selecting all basins/outlets and bringing up the respecting output control dialog. Saving the *TR-20* file and running it will then create the ".THY" file.

- *WMS* will not read in *TR-20* files with more than one COMPUT record. Only one COMPUT record will be read in, and it will be the last COMPUT record in the file. If you need to make more than one computation, simply change the necessary input parameters in *WMS*, save the file, and run *TR-20* again.

- *WMS* will not read in data from ALTER, DELETE, INSERT, IPEAKS, PEAKS, or DURINC records.

- *WMS* adds basin, reservoir, outlet, reach, and diversion names in columns 73 through 78 of *TR-20* files. These names are not necessary to run *TR-20* or to read existing *TR-20* files into *WMS*.

- If you run into any other problems reading *TR-20* files, please contact the distributor you purchased *WMS* from.

1.6 Considerations for Reading Existing HSPF Files

While WMS is capable of reading HSPF User's Control Input (*.uci) files created in other programs or manually, some limitations exist. Some of these limitations are permanent in WMS, other are currently under development and will be resolved.

- WMS will only allow input from one WDM file and output to one OUT file. If more WDM or OUT files are included in the FILES block of the *.uci file, they will be ignored.

- WMS currently reads the SCHEMATIC and MASS LINK blocks to determine the structure and connectivity of the watershed. The NETWORK block is not yet supported.

- If a particular land segment contributes to more than one reach/reservoir segment, that land segment will be copied to a new segment for each reach/reservoir. This can result in many new segments if the *.uci file contains several of these cases. Each new segment created will have the exact same attributes as the original, and will represent the area of the original segment which contributes to each reach/reservoir.

- WMS currently does not read the COPY or GENER block from the *.uci file.

- The SPECIAL ACTIONS block is not yet supported.

Many of these limitations will be eliminated as development of the HSPF interface continues.

CHAPTER 2

General Tools

2.1 Introduction

This chapter is designed to familiarize you with the basic operations and layout of the *WMS* user interface. The interface to *WMS* has been designed in a modular fashion. Six separate modules representing different types of data are supported. These modules are briefly described in the first chapter. As you switch from one module to another, a portion of the interface (menu commands, tools, etc.) changes and a portion of the interface remain unchanged. The part that remains the same provides access to general tools that are used by all of the modules. These tools are described in this chapter. Tools that are specific to a certain module are described in later chapters.

2.2 *WMS* Screen

The *WMS* screen is divided into five main sections (when all windows are displayed): the *Graphics Window*, the *Hydrograph Window* (by default this window is not displayed), the *Tool Palette*, the *Edit Window*, the *Help Strip*, and the *Menu Bar* (Figure 2-1).

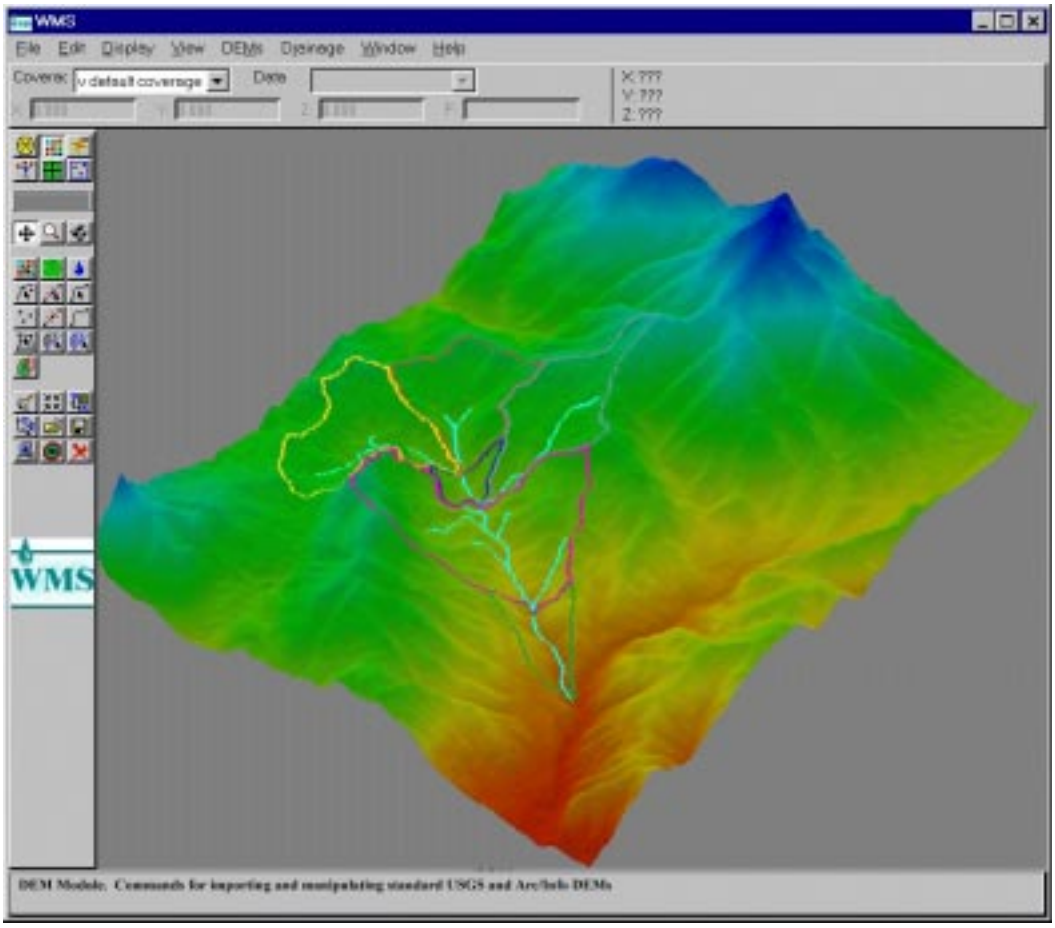


Figure 2-1 WMS *Screen.*

2.3 Graphics Window

Except for hydrographs, all graphically displayed data in *WMS* is displayed in the large graphics window in the center of the *WMS* application window. Graphical selection and interaction with TINs, DEMs, feature objects, hydrologic modeling trees (in previous versions the tree was displayed in a separate window, but in version 6.0 and higher it is displayed in conjunction with other graphical objects in the main graphics windows) is done by choosing the appropriate tool and then selecting in the graphics window. The action taken when performing a selection always depends on which tool is selected.

2.4 Hydrograph Window

The *Hydrograph Window* is used to display detailed two-dimensional plots of computed hydrographs, storage capacity curves, and other data results. When one of the supported hydrologic models is used, computed results are read and

displayed as small icons in the *Graphics Window*. These hydrographs can then be selected by their icons and displayed in more detail in the *Hydrograph Window*. The *Hydrograph Window* is a floating window and by default is not open. The *Show Hydrograph Window* command in the *Display* menu is used to toggle the window open and closed. It can be resized and moved to any location on the screen.

2.5 Tool Palette

The *Tool Palette* is divided into four parts as shown in Figure 2-2: modules, Background color and static tools, dynamic tools, and macros.

2.5.1 Module Palette

The *Module Palette* is used to switch between modules. Only one module is active at any given time. However, the data associated with a module (ex. a DEM) is preserved when the user switches to a different module. Activating a module simply changes the set of available tools and menu commands.

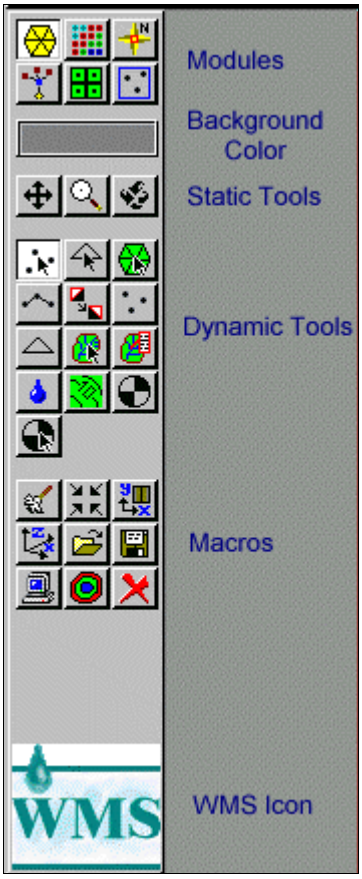


Figure 2-2 Professional Tool Palette.

2.5.2 Static Tool Palette

The tools that are available in every module are located in the *Static Tool Palette*. These tools are used for basic operations such as panning, zooming, and rotating. The static tools are as follows:

Backcolor Button

The *Backcolor* button is used to change the color of the background of the *Graphics Window*. Clicking on this button brings up a palette of standard colors to choose from.

Pan Tool

The *Pan* tool is used to pan the viewing area of the *Graphics Window*. When the *Pan* tool is active, clicking the mouse in the *Graphics Window* has the following results:

- If a point is clicked, the viewing area is shifted so that the point clicked corresponds to the center of the window.
- If the cursor is dragged while holding the mouse button down, the viewing area is shifted to simulate moving the image the direction and distance specified by the line defined while dragging the cursor. The image isn't updated until the mouse button is released.

Zoom Tool

The viewing area can be magnified/shrunk using the *Zoom* tool. When this tool is active, the following actions can be used to redefine the viewing area of the *Graphics Window*:

- A rectangle can be dragged around a portion of the display to zoom in on a particular region. The display is refreshed and the area inside the rectangle is expanded to fill the entire screen.
- If a point is clicked, the display is zoomed in around the point by a factor of two.
- If a point is clicked while the *SHIFT* key is held down, the display is zoomed out about that point by a factor of two.

Rotate Tool

The *Rotate* tool provides a quick way to rotate the image on-screen about the x and z axes. Two rotation methods are available:

- Holding down the mouse button and dragging the cursor in the *Graphics Window* rotates the object in the direction specified once the mouse button is released. A horizontal movement rotates the image about the z axis. A vertical movement rotates the image about the x axis. The amount of rotation depends on the length the cursor moves while the mouse button is down. The image isn't updated until the mouse button is released.

- Holding down both the *CTRL* key and the mouse button while dragging the cursor in the *Graphics Window* causes the rotating image to be updated dynamically.

See the View section later in this chapter for more information about changing the viewing angles.

2.5.3 Dynamic Tool Palette

When the active module is changed, the tools in the *Dynamic Tool Palette* change to the set of tools associated with the selected module. Each module has a separate set of tools. These module-specific tools are documented in their respective chapters.

Selection Tools

Many of the module-specific tools in the dynamic portion of the *Tool Palette* are selection tools (tools used to select objects such as triangles or vertices). For many commands it is necessary to first select some objects before issuing the command. For example, to delete a set of triangles in the TINs module, the *Select Triangles* tool is chosen, the set of triangles to be deleted are selected and the *Delete* command is selected from the *Edit* menu.

Most of the selection tools follow a standard selection protocol. Single items can be selected by clicking on the item. With this method, only one item can be selected at a time. When a new item is selected, any other currently selected items are unselected.

In many cases, multiple items need to be selected. If the *SHIFT* key is held down while clicking on individual items, the items are added to the set of selected items. A previously selected item can be unselected by holding down the *SHIFT* key and clicking on it again. This removes the item from the set of selected items without affecting other selected items. Multiple objects can also be selected by dragging a box around the items to be selected, or by choosing the *Select All* or *Select With Polygon* commands from the *Edit* menu.

You may clear the selection list at any time by clicking on a portion of the graphics window where no objects exist. This effectively clears the selection

list because whatever is currently selected becomes unselected and since you click in a location where no objects exist nothing is placed in the selection list.

2.5.4 Macros

Many of the more frequently used menu commands can be accessed through the macro buttons in the lower part of the *Tool Palette*. If your computer screen does not have high resolution (a large amount of screen display) capabilities, all or part of the macro buttons may not be visible. In such cases you can still use their equivalent menu commands. Some of the commands that have associated macros include display options, open file, save file, refresh, and frame. They are documented in more detail later in this chapter.

2.6 Edit Window

There are two lines in the *Edit Window*: the coverage and data selection boxes, and the coordinate edit boxes.

2.6.1 Coverage and Data Set Selection

The coverage selection combo box allows you to quickly specify the active GIS coverage. The active coverage is the only one that can be edited. Other operations such as drainage data computation require that the active coverage be the drainage coverage. Whenever you select a coverage from the combo box it becomes active and is equivalent to opening the *Coverages…* dialog from the *Feature Objects* menu and setting one of the available coverages to be active.

The Data set selection combo box allows you to change the currently active data sets. In the current version of WMS data sets are applicable only for scatter points and grids.

2.6.2 Coordinate Edit Boxes

The coordinate edit boxes are on the bottom of the *Edit Window*. The first three edit fields are used to edit the coordinates of a selected TIN vertex, feature point, or scatter point. Coordinates are changed by typing in new values and hitting the *ENTER* or *TAB* key. The fourth edit box is used to edit dataset values of selected scatter points.

2.6.3 Help Strip

The *Help Strip* is at the bottom of the *WMS* application window and is used for user prompts and to display context sensitive help messages. Some commands require selection in the middle of execution, at these times WMS rings a bell

and a prompt appears in the *Help Strip* with specific instructions. Once the instructions have been completed the prompt message is removed. Context sensitive help messages appear in the *Help Strip* as the cursor is moved over tools, macros, menu items, or dialog items.

The column of numbers that appear at the end of the *Help Strip* are used to display the coordinates and current function value of a grid, under the cursor when the displayed data are in plan view. The z coordinate corresponds to an interpolated value from the active TIN, DEM, or grid (depending on the active module).

2.7 Menu Bar

The commands in *WMS* are accessed through pull down menus located in the menu bar. Each menu can be accessed with the mouse or by holding down the *Alt* key and pressing the underlined letter in the menu title. Once a menu is visible the individual commands can be selected with the mouse or by holding down the *Alt* key and pressing the underlined letter in the menu command.

When the active module is changed, the menus change to a set of menus associated with the selected module. The first four menus (*File, Edit, Display, View*) are the same for every module. The remaining menus are dependent on the selected module.

If a menu item is dimmed, then it can not be used until the proper conditions exist. Typically this means the appropriate data has not yet been created or selected.

2.8 File Menu

The *File* menu is one of the standard menus and is available in all of the modules. The commands in the file menu are used for file input/output for the basic data types, for printing, and to quit the program.

2.8.1 WMS File Types

The file types supported by the *Open* command in the *File* menu correspond to the *WMS* data types only. The file input/output commands associated with specific analysis codes such as *HEC-1* are found in the menus associated with the analysis codes. Other supporting data such as *ArcView* shapefiles, DXF files, USGS DEMs, etc. are opened using the *Import* command discussed later in this chapter.

The basic file types are as follows:

TIN File File containing a set of vertices and triangle connectivity information.

DEM File File containing a rectangular portion of elevation points.

Map File File containing feature point and feature arc definitions.

Tree File The topologic data structure created as part of the lumped parameter models supported by WMS (HEC-1, TR-20, rational method, etc.) is stored in the tree file. The tree file also stores all of the modeling parameters associated with the hydrologic models, such as curve numbers, precipitation, etc.

Grid File File containing 2D finite difference grid information.

Scatter Point File File containing information for scatter points used in interpolating values to grids.

Data Set File File containing function or data set values which can be used in conjunction with scatter point or grid files.

Image File File containing the name of a pre-registered TIFF file along with the accompanying registration points.

Super File File containing a list of names of other files. This file is used to group a set of files together so that several files can be loaded at once by selecting the appropriate super file. DXF file and image file names can also be saved in the super file.

All of the *WMS* files are ASCII text files. The first item in each of these files is a keyword signifying the file type. The formats for these files are described in Chapter 20.

2.8.2 New

The *New* command deletes all data associated with all data types and all modules. This command should be selected when an entirely new modeling project is started.

2.8.3 Open

The *Open* command is used to read in one or more of the *WMS* file types. This command brings up a file browser from which a single file is selected. The program reads the keyword at the beginning of the file to determine the type of file selected and the appropriate routine is used to input the file. The

default file extension will change (i.e. *.tin, *.dem, *.map, etc.) depending on the module you are in. However, you can change the filters and/or open any *WMS* file type no matter which module you are currently in. If a super file is selected, all of the files listed in the super file will be read in. Files, other than the *WMS* file types, can be read using the *Import* command discussed later in this chapter. The *Open* command has a macro in the tool palette .

2.8.4 Save As

The *Save As* command is used to save the *WMS* file types to disk. This command brings up a dialog that contains a check box, and a filename for each of the *WMS* file types (See Figure 2-3). If a data type does not currently exist in memory, the check box and filename for that data type is dimmed. The check box can be toggled off or on to indicate whether or not the data type is to be saved to disk. At the top of the dialog the current path name is displayed. The File Browser button can be used to change directory paths and the name of the super file. A prefix for all files can be entered into the edit field and then updated to all files using the adjacent button. The current display settings may also be saved as part of the super file so that when the file is re-opened the display settings will appear just as they were when the files were saved. A description of these settings is given in the next section. The *Save As* command has a macro in the tool palette .

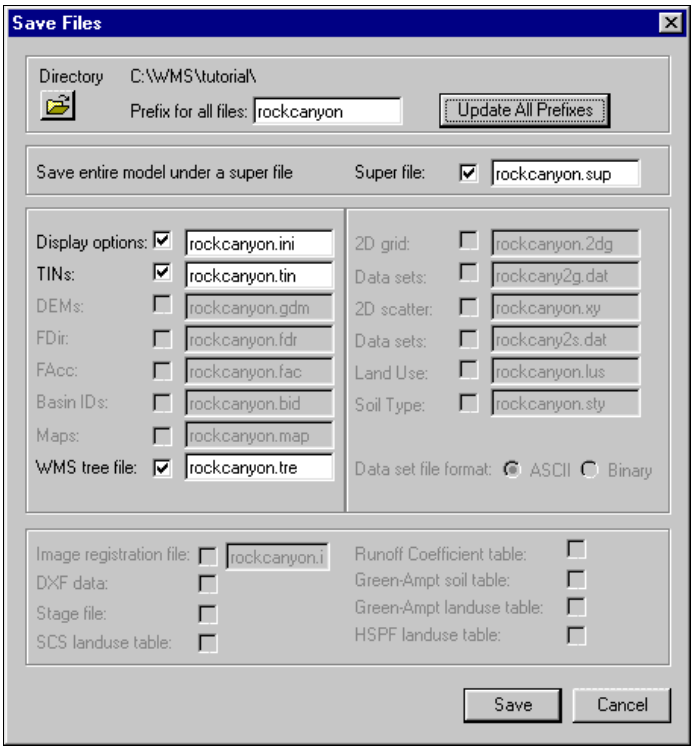


Figure 2-3 Save Files Dialog.

2.8.5 Save Current Settings

The *Save Current Settings* command is used to save the current display settings of the program (display options, defaults, etc.) to a default settings file. *WMS* opens the default settings file each time it is launched and initializes the settings to the defaults stored in the file. A local copy (specific to a project) of these settings can be saved and then opened within a super file as well.

2.8.6 Import File

Several different file types can be imported into *WMS* for use in creation of feature objects, grids, and TINs. These files (not native to *WMS*) can be opened using the *Import* command in the *File* menu. The dialog shown in Figure 2-4 is used to specify which file is to be imported, and the following paragraphs give a brief description of the different files and what they can be used for.

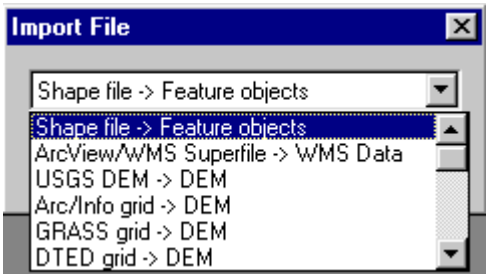

Import File
Shape file -> Feature objects
Shape file -> Feature objects
ArcView/WMS Superfile -> WMS Data
USGS DEM -> DEM
Arc/Info grid -> DEM
GRASS grid -> DEM
DTED grid -> DEM

Figure 2-4 Import Dialog.

ARC/INFO® & *ArcView*® Shape Files

ARC/INFO® or *ArcView*® shape files provide the easiest method to import GIS data into *WMS*. Unfortunately the shape file format is extremely redundant, meaning that points or lines that are shared by lines or polygons are multiply defined. Therefore, the first time you import a shape file it can take up to several minutes (depending on size) to build the correct line or polygon topology. If you intend to use the data from the shape file in more than one session you should save it as a *WMS* map file after importing the first time. Further, after importing the shape files you may wish to consider the following:

1. *Clean* (see page 3-27) the feature objects in order to snap nodes within a certain distance, intersect arcs, and eliminate dangling arcs.

2. *Reorder Streams* (see page 3-28) for arcs which will be used as stream arcs. *WMS* requires that the direction of an arc (from-node to to-node) be from “downstream” to “upstream.”

3. *Build Polygon* (see page 3-33) so that *WMS* can define the topologic tree used for hydrologic modeling. After intersection of arcs, reordering of streams, etc. it is often necessary to rebuild the polygon topology so that the topologic structure is consistent with the tree used for hydrologic modeling.

More information about how shapefile data can be used to define watersheds is given in the introductory chapter on page 1-21. The same dialog is used to import point, line, or polygon shape files and is shown in Figure 2-5.

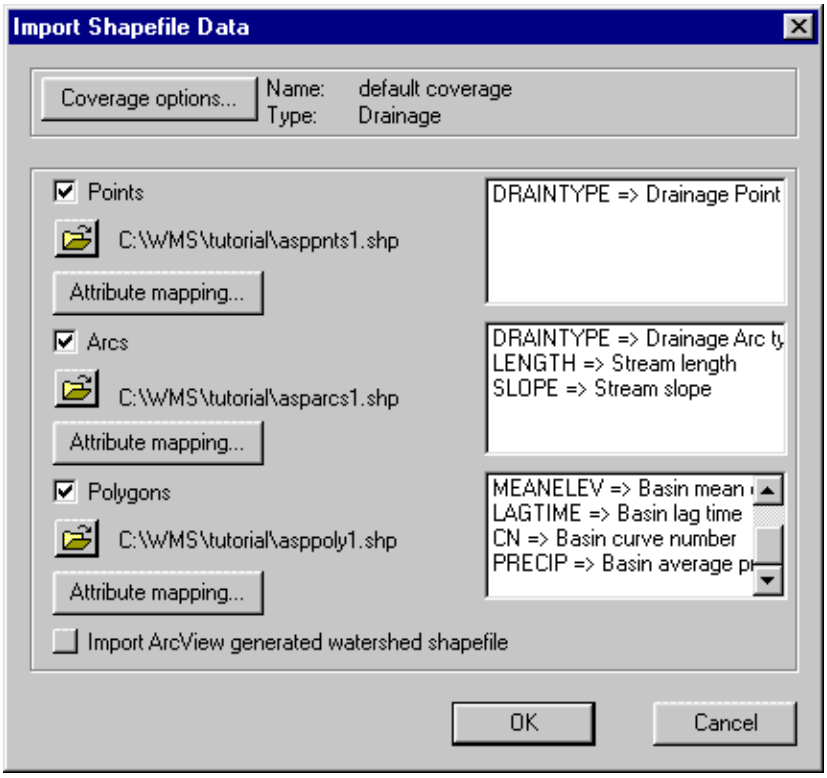


Figure 2-5 Import Shapefile Data Dialog.

The following key words are used to automatically map shapefile (dbase or .dbf) attribute field names to data within *WMS*.

Table 2.1 Keyword names to map Shape file attribute item names to *WMS* variables.

Parameter	Name	Description/Possible values	
Point Attributes			
Type of point	draintype	0 = generic 1 = link break 2 = weir 3 = bridge 4 = culvert 5 = outlet point	*
Arc Attributes			
Drainage arc type	draintype	0 = generic 3 = general stream 4 = highway	*
Drainage Manning's n	dmannings	Manning's n-value (floating point)	
Polygon Attributes			
Drainage polygon type	draintype	Drainage polygon type: 0 = generic 1 = boundary 2 = lake	*
Drainage basin id	basinid	Drainage basin id (integer)	
sub-basin area	basinarea	Basin area (float)	
sub-basin slope	basinslop	Average slope within the sub-basin (float)	
sub-basin maximum flow distance	mfdist	Max flow path, including overland and stream flow (float)	
sub-basin max flow distance slope	mfdslope	Slope along the max flow path as defined above (float)	
sub-basin distance to centroid	centdist	Distance from centroid to closest point on main channel (float)	
sub-basin stream centroid to outlet	centout	Distance from point in stream closest to centroid to outlet (float)	
sub-basin slope from centroid to outlet	slcentout	Slope along the distance defined above (float)	
sub-basin percent southfacing	psouth	Percentage of area facing south, 0.0-1.0 (float)	
sub-basin percent northfacing	pnorth	Percent of area facing north, 0.0-1.0 (float)	
sub-basin maximum stream length	mstdist	The longest stream distance within the basin (float)	
sub-basin maximum stream slope	mstslope	The slope along the longest stream distance (float)	
sub-basin length	basinlen	Distance to furthest point along basin perimeter (float)	
sub-basin shape factor	shapefact	Basin length divided by basin area (float)	
sub-basin sinuosity factor	sinuosity	Maximum stream length divided by basin length (float)	
sub-basin perimeter	perimeter	Perimeter of basin (float)	
sub-basin average elevation	meanelev	Average elevation (float)	
sub-basin centroid	centroidx	Basin centroid, closest point in basin if centroid is outside of the basin (X-coord) (float)	
sub-basin centroid	centroidy	Basin centroid, closest point in basin if centroid is outside of the basin (Y-coord) (float)	

sub-basin name	basinname	Basin name (string)	
sub-basin lagtime	lagtime	Lag time, in Hours (float)	
sub-basin time of concentration	tc	Time of Concentration, in hours (float)	
sub-basin SCS curve number	cn	SCS Curve number computed from hydrologic soil type and land use (Integer)	
sub-basin average precipitation	precip	Basin average precipitation, in inches (float)	
landuse	lu_code	Land use code from the SCS land use table. Possible values range from 0-127	
soil type	hydgrp	SCS Soil type, A, B, C, or D or 0, 1, 2, or 3	
runoff coefficient, C	runoffc	Rational method runoff coefficient, C (float)	

* means this is essential to import into *WMS* and create a watershed model directly. The three essential items are point, arc, and polygon types. The general stream arc should be used to represent a stream in a watershed model. The boundary polygon type should be used to represent a polygon boundary. The outlet point type should be used to represent a watershed outlet or sub-basin outlet point.

In order to import shapefile attributes into *WMS* and build a tree automatically, the following conditions must be met:

1. A point coverage containing watershed and sub-basin outlets with the appropriate type (outlet point) attribute defined must exist.

2. An arc, or line, coverage containing streams in the watershed with the appropriate type (general stream) attribute defined must exist.

3. A polygon coverage containing watershed boundaries must exist.

4. There cannot be any overlapping arcs.

5. Stream arcs must be created from the downstream to the upstream node for all stream arcs.

If a data value in the shapefile corresponds to a *WMS* variable but it is not defined with the appropriate keyword it can be mapped manually using the *Attribute Mapping* dialog shown in Figure 2-6. One item from the database fields window is selected and the corresponding coverage attribute field is also identified. Finally, the *Map* button is selected to define a new mapped attribute. The *Unmap* button can be used to remove a pair of mapped fields.

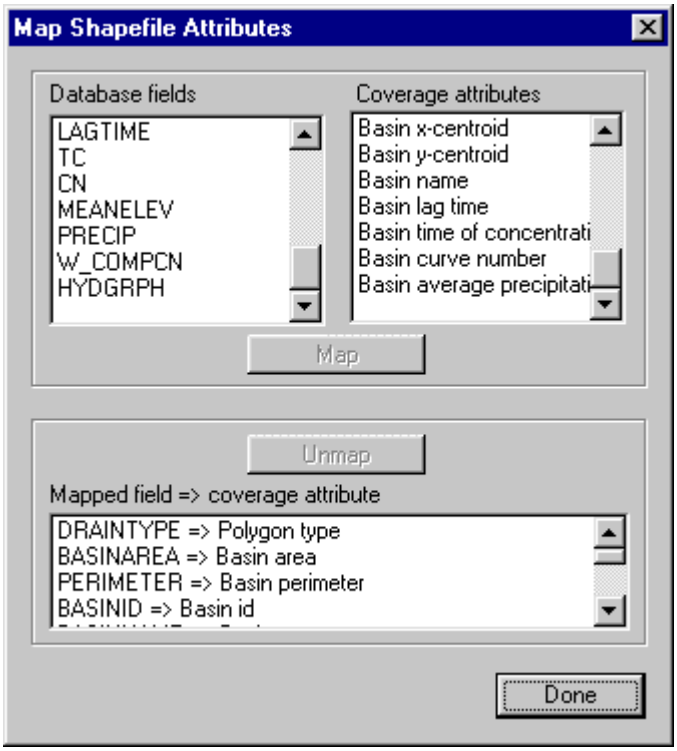


Figure 2-6 Map Shapefile Attributes Dialog.

ArcView/WMS Superfile

Besides the tools in *WMS* for editing data imported from shapefiles, a special extension for *ArcView*® has been created which allows you to create the outlets, streams, and basins themes, reorder streams, rename attributes, and then export to a *WMS*/*ArcView*® super file which will allow you to open all three themes by importing the super file. This extension is placed in the *wmshydro* directory (under the main *wms* directory) when installing *WMS*. It is also available on the *WMS* website and has a separate document describing its usage. In order to be activated in *ArcView*® you must move the extension file (.avx) to the Ext32 directory found under the *ArcView*® installation directories.

Streams as Feature Points

The *Streams to Feature Points* option can be used to read in a file of xy (or xyz) points defining a stream and convert them to feature points. These files can be created using any digitizing software, but the format of the file must be as is shown in section 20.14.

DLG Files as Feature Arcs

The *DLG to Feature Arcs* option allows a Digital Line Graph file to be imported and points connected into a series of arcs. USGS DLG files, like

DEMs, can be downloaded via the Internet (see the introduction to the DEMs module) and are available for many parts of the U.S..

In addition to streams, DLG files contain points for highways, canals, railroads, and many other linear map features. When importing you may select whether to import all points, or only those points which are flagged as stream points. You may also decide whether or not consecutive points should be linked together as arcs. DLG files often contain more data than is necessary for creating models in *WMS*. Typically they are useful for getting a good idea of where streams, roads etc. are. However, often their best use is as a background map which can be used as guide for manual creation/placement of streams and roads.

When choosing this option a second dialog will appear which allows you to "add" several adjacent DLG files and then import them all at the same time.

ACAD Streams as Feature Arcs

The *ACAD Streams to Feature Arcs* option can be used to read in an xy (or xyz) file similar to the one documented in the previous paragraph in and create a series of feature arcs. This file is exported by some AutoCAD applications, and the format is documented in section 20.15. This format can be used to conveniently convert any xy(z) data to a file which not only reads the points, but also connects them together. Since *WMS* now supports direct importing and conversion of polylines from DXF files, AutoCAD users will typically find this method easier (see Section 3.5.2).

USGS, SDTS, *ARC/INFO*® ASCII, DTED, and GRASS DEMs (Grids)

The USGS and other government and proprietary agencies have distributed both the 1:250,000 and 1:24,000 scale USGS digital elevation files in a USGS-defined format for a number of years. The 1:250,000 DEMs available for download from the USGS web site remains in this format. However the 1:24000 DEMs are now distribute on this site in the SDTS format. Other common DEM file formats include *ARC/INFO*® ASCII Grid, DTED, and GRASS. All of these dialogs are imported in *WMS* in essentially the same way as described below, but you will need to know which format the DEM is in prior to reading the DEMs for use in *WMS* (particularly in the case of the commonly used USGS 1:24000 DEMs that are available in both the USGS and SDTS formats). The DEM file type is specified in the *Import File* dialog (see Figure 2-4) and include all five of the previously mentioned formats (the USGS DEM option refers to both the older single file format and the newer SDTS format). Once you specify the format type the *Import DEMs* dialog controls file selection as well as other import options. For example, multiple files of the same format may be "tiled" together when importing, but you cannot mix and match between two or more different formats. **NOTE:** *ARC/INFO*® grids that are to be imported must be saved as ASCII files from *ARC/INFO*® in either feet or meter units and not latitude-longitude.

The *Import DEMs* dialog is used to examine the limits of DEM files as well as defining a clipping boundary to eliminate regions outside the area of interest prior to actually reading the elevations in. The *Add* button is used to add a new file to the list of files that will be read. The standard file opening dialog appears from which you may select DEM files. Once the file is added to the list a bounding rectangle is displayed in the small graphics window in the center of the dialog. As additional files are added the graphics region is updated with new rectangles in order to provide an understanding of where DEMs are located in relation to one another.

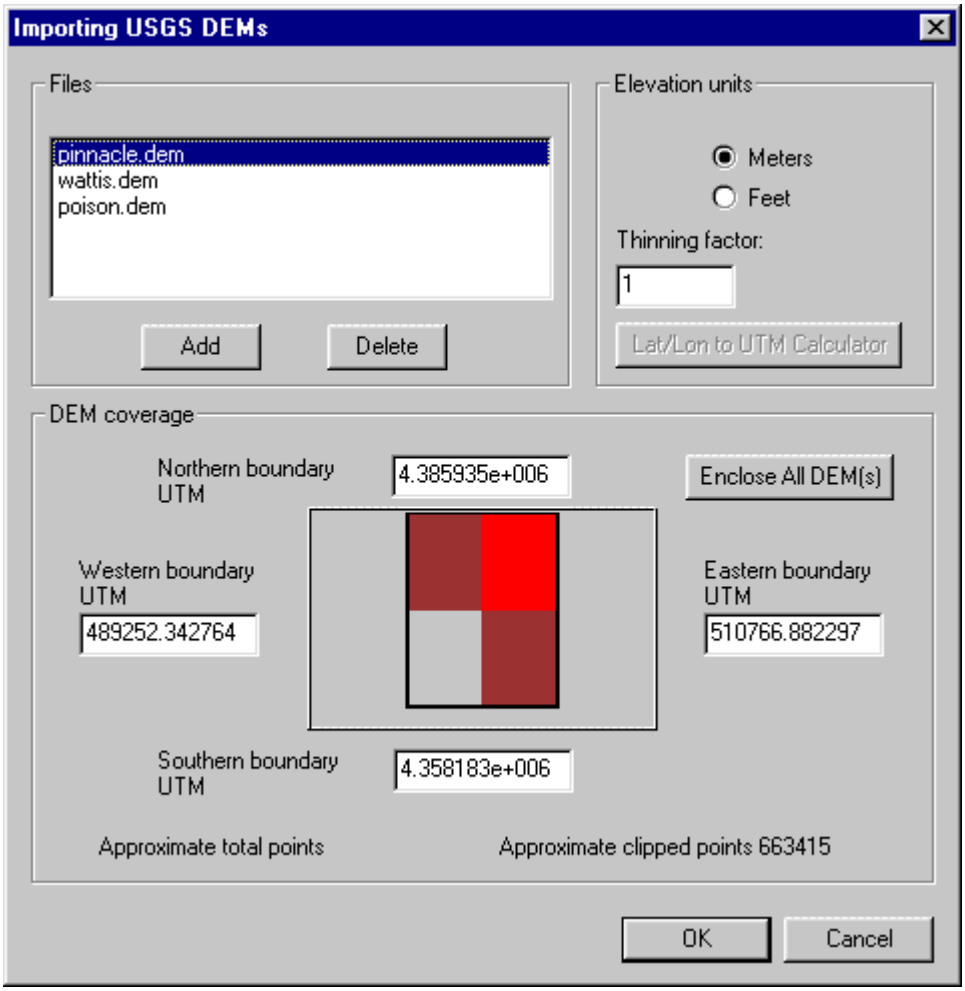


Figure 2-7 Import DEMs Dialog.

A small black rectangle is displayed in the central graphics window. Only elevation points inside this rectangular region will be read in when hitting OK from this dialog. This boundary rectangle can be modified in three different ways.

1. DRAGGING - Using the mouse you can click near one of the four edges of the bounding rectangle and drag it to a new location. If you

click near a corner both edges will be adjusted. If you click in the center of the rectangle then the entire rectangle can be translated to a new location. As you drag edges to new locations their corresponding values are automatically updated in the Edit Fields.

2. EDIT FIELDS - Any one of the western, eastern, northern, or southern boundaries can be explicitly set by changing the values in their corresponding edit fields. As new values are entered the display in the small graphics window can be updated by tabbing or by clicking the cursor outside the current edit field.

3. ENCLOSE ALL DEMs - This button can be used to force the edges of the bounding to rectangle to correspond to the limits of the DEM files which have been added to this point. By default when a new DEM file is added the bounding rectangle is adjusted to enclose all DEMs.

The thinning factor can be used to reduce the number of elevation points read. A thinning factor of 2 means that every other row and column would be read, reducing the number of total points by a factor of 4. A factor of three means that every third row and column would be read reducing the total by a factor of 9, etc.

The elevation units toggle can be used to specify whether imported DEM points have meter or feet for units of elevation. If a DEMs base elevation units are feet and the toggle specified meters, all elevations are converted when reading. This is particularly important when trying to read two adjacent DEMs with different base elevation units.

NOTE: This option does not change the base planimetric units of the DEM and you should make the elevation units consistent with the planimetric units in order to ensure that slopes are computed properly when computing basin geometric parameters.

At the bottom of this dialog the total number of DEM points from all DEM files which have been added and the approximate number of points inside the bounding rectangle are displayed. These numbers can be used to determine how many points your system is capable of reading. For example, each DEM point requires 5 bytes of memory, so that if you read an entire 1:250,000 DEM with about 1.4 million points, 1.4 meg * 5 bytes = 7 meg of memory would be required (in addition to whatever other memory being used by *WMS*). This means that you would need at least 8 meg of RAM, or some type of virtual memory capabilities would be required to read in the entire DEM file.

This same dialog is used for all five types of DEMs supported in the *WMS* import options: USGS, *ARC/INFO*®, GRASS, DTED, and SDTS.

DEMs in Different UTM Zones

Occasionally two adjacent USGS DEM files will be read in but do not appear adjacent in the import dialog. This occurs because while the two DEMs are adjacent, they lie in different UTM zones. X coordinates within UTM zones repeat and therefore the DEMs do not lie adjacent to one another as they should. *WMS* does not contain the utility functions necessary to transform a DEM from one UTM coordinate zone to another. However, *ARC/INFO*® and possibly other GIS software can be used to convert from one zone to another. Since *WMS* can import *ARC/INFO*® grids the DEMs can be merged, projected to consistent zones, and then exported as an ASCII grid file which can be imported into *WMS*.

Gridded Attributes -> *WMS* Attributes

Besides an elevation, DEM points may contain certain attributes such as a basin id, flow direction, flow accumulation, and stream id. In addition land use id's and soil type id's used in computing composite curve and runoff coefficient values (see section 15.2) are also stored as DEM attributes (also called grid cells). These attributes are imported using the DEM Masks -> DEM Attributes option. Figure 2-8 shows the different options for importing DEM attributes. You must also specify whether the file format is *ARC/INFO*®, GRASS, or TOPAZ. Once the attribute and file type options are chosen, you must then find and open the file containing the attributes.

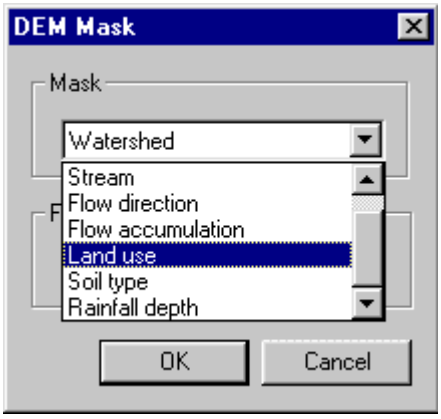


Figure 2-8 DEM Mask Dialog.

GIS Grids as 2D Grids

The *ARC/INFO*® and *GRASS Grids* to *2D Grids* command can be used to read either an *ARC/INFO*® or GRASS ASCII grid file. The file is read and a grid created in *WMS* which conforms to the imported grid. The ASCII values for the grid cell centers are used to create the elevation function for the grid. Such grids can then be visualized using *WMS*'s tools for grids (see Chapters 16 and 17) or used to create CASC2D finite difference models (see Chapter 19).

ARC/INFO® TIN to TIN

An *ARC/INFO*® TIN can be imported if it has been exported from *ARC/INFO*® using the UNGENERATETIN command with the *NET* option. This particular format saves node (vertex) coordinates, a list of edges and a list of triangles, making it possible to restore the TIN topology in *WMS* exactly as it was in *ARC/INFO*®. Once imported it can be used to perform all of the watershed characterization operations available in *WMS*.

Tabular Data to Scattered Data

Tabular data can be generated from a common spreadsheet (or other database management) program and read into *WMS* to create a set of scattered points with either a single or a transient data set.

Tabular Data to Feature Object Data

Tabular data from *ARC/INFO*®, or other database program, can be imported into *WMS* and associated with feature objects. Currently only land use and soil type data is supported. The file consists of a self-describing header followed by the data to be imported. The self-describing header tells *WMS* what values are represented, and the order of the data in the file.

The following are the available key words that should appear on the first line of the file you wish to input. The key words can be in any order, but *WMS* will interpret the data on the following lines in the order that the key words appear. One of the feature object ID cards must be included as one of the key words. If data exists in your tabular data that cannot be associated with one of the feature objects and you don't want to edit it out of the file you can use the NODATA key word for that column(s).

POINTID, ARCID, POLYID, LANDUSE, SOILTYPE, NODATA

The following is an example tabular data file which associates a land use id with polygons.

```
POLYID LANDUSE
1 1
2 23
3 29
4 30
5 28
6 24
```

Spaces, commas, or tabs can be use to delimit the different fields of the data file.

NEXRAD Precipitation -> Scatter Points

This option was developed by personnel at the Waterways Experiment Station (WES) for importing "in-house" formatted files of NEXRAD precipitation. If you have access to data from WES contact us for what needs to be done to import this into *WMS*.

The *WMS* developers would like to be able to support more standard formatted files of NEXRAD data. If you have additional information please contact the developers for help in implementation.

Rain Gage File -> Scatter Points

The rain gage file supported is the CASC2D rain gage file. These files contain gage information for long-duration events and can be converted to scatter points for visualization purposes.

TIFF Image Data

A tiff file can be imported and registered so that it appears as a "backdrop" in *WMS* or mapped to TIN data when shading. More details on importing TIFF image data are given in section 3.4.2.

DXF Vector Data

DXF files may also be imported and then converted to feature objects, TINs, or simply used to enhance the display of a project. More information on importing DXF files is given in section 3.5.

XYZ Data as TIN Vertices

Choosing this option allows you to read in a space-delimited file containing x, y, and z coordinate values. The *Triangulate* command in the *TINs* menu can then be used to create a TIN from the xyz data. The file must be space delimited in order for *WMS* to import it.

2.8.7 Export File

The *Export File* command is used to export objects from *WMS* to files that can be used with other applications. A number of popular file formats are supported. Once one of the formats is selected and the OK button is selected, you choose a name for the file using the standard file browser.

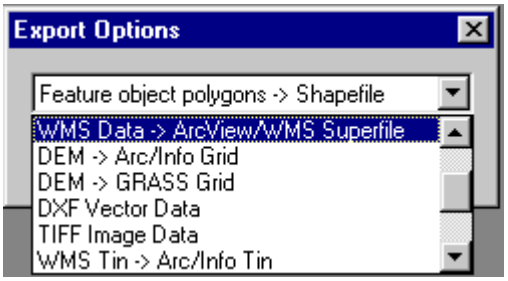


Figure 2-9 Export Dialog.

Feature objects

ArcView® requires that polygons, lines (arcs), and points be defined in separate layers or themes. Therefore, you must export each of these data types in *WMS* to separate shape files. Selecting the appropriate feature object export option (polygons, arcs, or points) does this.

If you want to export outlets, streams, and basins together you can save them to an *ArcView*®/*WMS* Superfile. However, in order to read them in together in *ArcView*® you will need the *WMS*Hydro extension for *ArcView*®.

DEM->ARC/INFO Grid

This option saves the currently defined DEM to an *ARC/INFO*® ASCII grid file. It can then be imported into *ARC/INFO*® or *ArcView*® (with spatial analyst).

DEM->GRASS Grid

This option saves the currently defined DEM to a GRASS grid file. It can then be imported into GRASS.

WMS TIN->ARC/INFO TIN

A TIN in *WMS* can be exported to the *ARC/INFO*® ASCII TIN file format. It can then be imported into *ARC/INFO*® or *ArcView*® (with 3D analyst).

2D scatter points -> point generate file

Export the active scatter point set as an *ARC/INFO*® point generate file.

Computed drainage data -> tabular data

Drainage data such as areas, stream lengths, and slopes can be exported or saved to a file using the *Export File* command and choosing the drainage data file type. This file is intended to be used in conjunction with a spreadsheet program and therefore the format can be specified as either comma, space, or tab delimited. Only the data whose display boxes are set in the *Geometric*

Attributes Options dialog (accessed from within the *Drainage Display Options*) are written to the file.

Drainage boundaries & streams -> DLG

A DLG or digital line graph is a generic file format which can be interpreted by most common GIS programs. Only streams and polygonal boundaries representing drainage basins can be saved to a DLG file. Careful attention must be paid to the referencing points written to the DLG file, as they will have to be edited in order to reference this file to another coverage being used within the GIS program.

DXF file

Export the current the geometry defined in the current *WMS* scene as a DXF file.

TIFF file

Export the current data displayed in the *WMS Graphics Window* as a TIFF file. The image can be exported as either a shaded image or a wireframe image.

2.8.8 Print

Printed copies of *WMS* window displays can be generated by using the print command. The UNIX version of *WMS* will create a PostScript file that can be sent to any PostScript printer. The UNIX version can also create encapsulated PostScript files that can be imported into many other programs. The MS-Windows version of *WMS* will print to any printer supported by Windows.

When the *Print* command is selected, a dialog (Figure 2-10) appears allowing you to change a number of printing parameters. The *Printer/Postscript Setup* button accesses the *Windows Printer Setup* dialog in the Windows version and the *Postscript Setup* dialog (Figure 2-11) in the UNIX version. The *Page Layout* button accesses the *Page* dialog (Figure 2-12). The *Display Options* button accesses the *Print Display Options* dialog. These dialogs are described below.

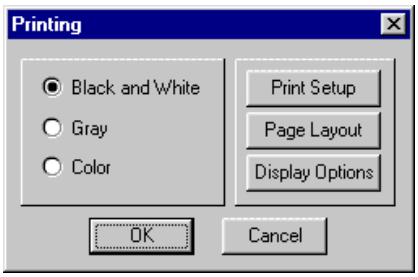


Figure 2-10 Print Options Dialog.

Printer/Postscript Setup

The *Printer/Postscript Setup* command allows you to control the orientation of the printed image on the sheet of paper and the paper size. The dialog shown in Figure 2-11 is for UNIX versions. For Windows versions the standard Printer Setup dialog for the currently selected printer device will be brought up which allows you to change other relevant parameters of the currently selected printer. For the Windows version, the current printer can be changed using the Windows Print Manager.

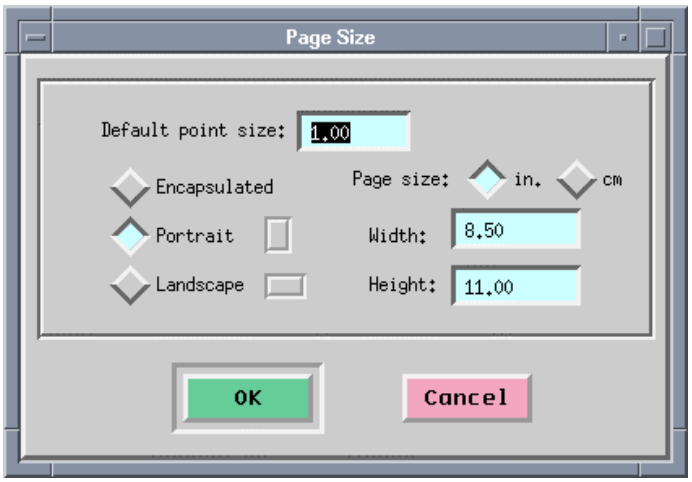


Figure 2-11 Page Size Dialog (UNIX only).

Page Layout

The *Page Layout* dialog allows you to change the size and position of the printed image on the paper. The image size is controlled in one of two ways:

1. By specifying the model units for an inch (or centimeter) of the paper. The scale toggle determines whether or not a scale legend is printed. The text color and font of the scale legend can be selected by clicking on the colored rectangle to the left of the toggle. If the *Maintain Aspect Ratio* toggle is not on then a scale legend cannot be printed.

2. By using the two scroll bars just under the page display. When the *Maintain Aspect Ratio* box is checked, moving one of the scroll bars will also move the other scroll bar. The current image size is displayed to the right of each scroll bar.

The *Center* button allows you to center the image on the page. The *Max Aspect* button sets the image to a size that will just fill the paper, maintaining the aspect ratio, with a 0.25 inch margin on either the left and right or top and bottom borders (depending on the critical direction and paper orientation).

At the bottom of the dialog, you can change the margins of the page. The margins can be referenced from the left, right, center, top, or bottom. The unit of measure is the same as the paper size.

The image may also be positioned on the paper by clicking on the box representing the image and dragging it within the paper display.

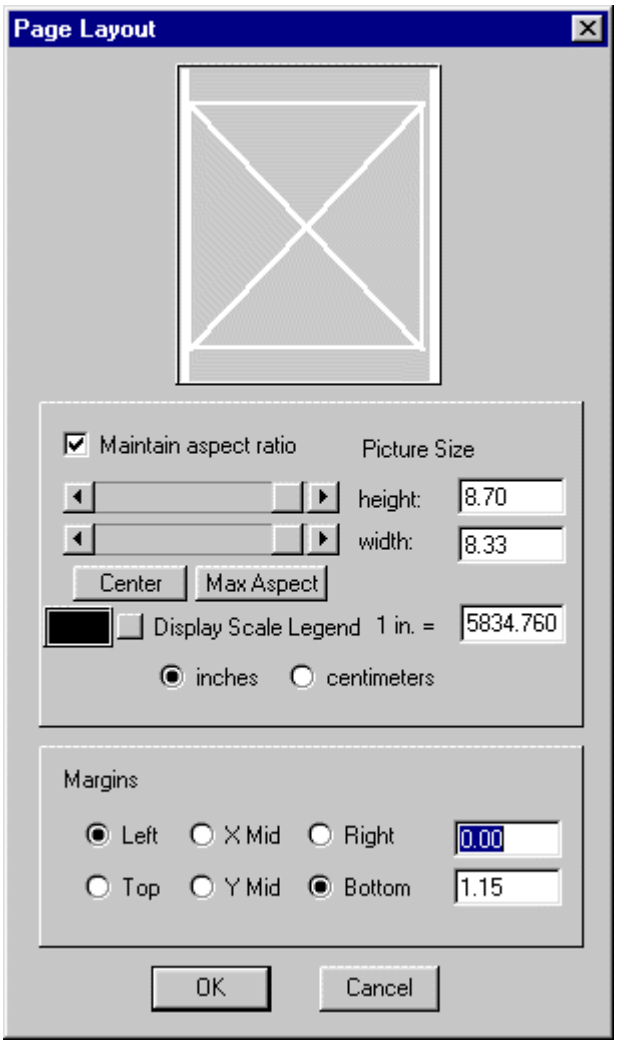


Figure 2-12 Page Layout Dialog.

Print Display Options

When the Print command is selected, the current display options are used to control what is printed. When an image is displayed in the *Graphics Window*, it is, by default, displayed as a wire-frame image. A continuous shaded image is displayed only when the *Shade* command is issued. The *Print Display Options* dialog allows you to specify whether the image will be printed as a

wire-frame, shaded image, or smoothed contours. It also allows flow patterns to be displayed in addition to the selected display mode.

2.8.9 Printing Files (UNIX Platforms)

Once the printing options are selected, the current image in the *Graphics Window* is printed by selecting the *OK* button from the *Print* dialog. With the UNIX version of *WMS*, the image is not sent directly to a printer, but is sent to a PostScript file. This file can then be sent to a printer from the UNIX command line.

2.8.10 Printing Files (PC Platforms)

When printing from the PC version of *WMS*, the image is sent directly to the printer according to the current settings of the standard Microsoft Windows printer controls. Most printers, particularly those which support postscript, allow you to save an image to a file rather than sending it directly to the printer. Such an option would be accessible through the *Printer Setup* command.

2.8.11 Viewing Files

The *View File* command allows you to examine any text file from within *WMS*. This command is particularly useful if errors occur while running a simulation using one of the hydrologic models supported by *WMS*. If a model does not run to a successful completion, errors can usually be found by examining the ASCII output file.

On PC computers the file is placed within the MS Windows Notepad (or WinWord depending on the file size) application to examine the file. On UNIX computers you will be prompted for the name of a UNIX editor (vi is the default) and then the file is brought up in the specified editor.

2.8.12 Get Info

The *Get Info* command brings up a dialog that reports basic information concerning the data of the current module. For example the number of vertices and triangles in a TIN is reported when the *TIN* module is active.

2.8.13 Demo Version

The demo version is disabled for the professional version. It is used from the basic version to toggle into a demo of the professional version.

2.8.14 Register

The *Register* command is used for enabling *WMS*. For information on how to register your copy of *WMS* consult your *WMS* installation guide or distributor.

2.8.15 Exit

The *Exit* command is used to exit the program.

2.9 Edit Menu

The *Edit* menu is one of the standard menus and is available in all of the modules. The commands in the *Edit* menu are used to select objects and delete objects.

2.9.1 Deletion Commands

The different ways to delete data in *WMS* makes it very easy to eliminate any portion or all of the data associated with a given model. One important thing to remember is that deleting in *WMS* does not cause the data to be permanently removed or erased from the hard drive, but rather eliminates it from core memory for a specific run of *WMS*. For example if you read a TIN in from a file and then delete the TIN it will no longer appear in *WMS*, but the file that was read in is not deleted from the hard disk.

The methods that can be used to delete data from *WMS* include:

1. Deleting all data and starting over with the *New* command in the *File* menu.

2. Deleting selected items (vertices, triangles, arcs, etc.) with the *Delete* command in the *Edit* menu.

3. Deleting all data associated with a given module (i.e. all TIN data in the TINs module) using the *Delete All* command in the *Edit* menu.

4. Deleting object specific data such as all feature objects, drawing objects, or DXF data with the *Delete* command found in individual menus.

Delete

The *Delete* command is used to delete any selected objects. This command is also equivalent to hitting the *DELETE* or *BACKSPACE* keys. The *Delete* command as a macro in the tool palette .

Delete All

The *Delete All* command is used to delete all of the data associated with the active module, whether or not they have been selected. It is similar to the *New* command in the *File* menu except that the *New* command deletes all data in all modules.

2.9.2 Selection Commands

Select All

The *Select All* command selects all items associated with the current selection tool, providing that the tool supports the *Select All* option.

Select With Polygon

The *Select With Polygon* command allows you to enter an irregular polygon enclosing the items to be selected (one of the selection tools must be active). To enter the polygon, click on both the polygon's starting point and each intermediate point defining the polygon and double click on the ending point. All items within the polygon will be selected. If an error occurs while entering a polygon, the following keys can be used:

- *BACKSPACE* or *DELETE* - Back up one line segment.
- *ESC* - Abort entering the polygon, and selection by polygon.
- CONTROL - Holding the CONTROL key down while moving the cursor causes all previously entered segments of the polygon to be moved simultaneously.

In addition to defining a polygon using the cursor, a polygon created in the *Map* module may be chosen to define a selection polygon.

2.9.3 Confirm Deletions

Whenever a set of selected objects is about to be deleted, the user is prompted to confirm the deletion. This is meant to ensure that objects are not deleted accidentally. Selecting the *Confirm Deletions* command in the *Edit* menu can turn off this option. The check mark in front of the command is present when this option is turned off and is not when it is turned off.

2.9.4 Copy to Clipboard (PC Platforms Only)

With Windows versions of *WMS*, the contents of the *Graphics or Hydrograph* windows can be saved as a metafile to the Windows clipboard by selecting the

Copy to Clipboard command. The clipboard can then be "pasted" into other Windows programs, such as word processors or drawing packages.

As with printing, *WMS* automatically determines which windows have data displayed and will prompt for the window to be saved to the clipboard when more than one has some type of data being displayed.

2.10 Display Menu

The *Display* menu is one of the standard menus and is available in all of the modules. The commands in the *Display* menu are used to control how attributes of a TIN, DEM, Feature Objects, etc., are being displayed, to set up a drawing grid, to control how contours are displayed, and to generate shaded images.

2.10.1 Display Options

New in version 6.0 all display options have been combined in a single a dialog that contains tabs to the separate option groups (formerly each tab was a separate dialog that was accessed from an appropriate menu). The *Display Options…* command in the *Display* menu is used to access the dialog shown in Figure 2-13.

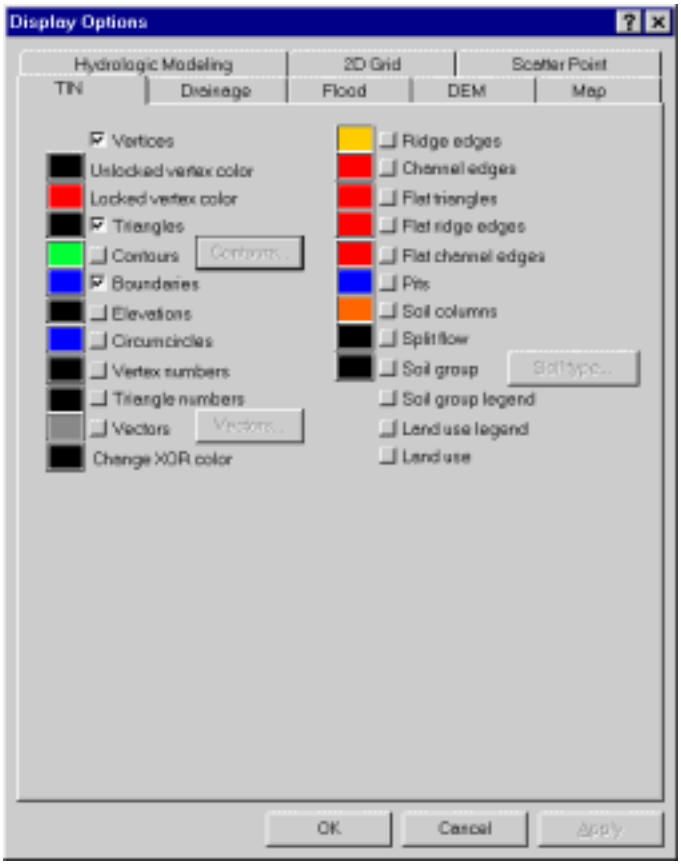


Figure 2-13 Display Options Dialog.

Selecting the check box to the left of the named option toggles on/off a display option. Color, style, thickness, and other attributes (the specific attributes depend on whether it is a point, line, text, or polygon attribute) can be modified by selecting the color box that is left of the toggle box.

Each tab represents a group of display options that are used in the different modules. When the Display Options… dialog comes up, the front tab will be determined based on the current module. However, display options for any tab can be set simply by selecting the desired one. Some modules have more than one tab to define all of the display options. A listing of which tabs are associated with each module is given below.

- TINs module - TIN, Drainage, Flood.
- DEMs module - DEM (includes DEM drainage display options)
- Map module - Map
- Hydrologic Modeling module - Hydrologic Modeling (includes tree display options and hydrograph display options)
- 2D Grid module - 2D Grid
- Scatter Point module - Scatter Point

The display options for each tab are discussed separately in the chapters describing the specifics of each module. The *Display Options…* command has a macro in the tool palette .

2.10.2 Contouring

TINs, DEMs, and grids can be contoured in *WMS* by turning on the contour option in their respective *Display Options* dialog tabs. When a TIN or DEM is contoured, elevations are used to generate the contours. When a grid is contoured the active data set is used to define contour values. See Chapter 11 for more information on how data sets are used.

The options used to generate contours can be edited by selecting the *Contour Options* command in the *Display* menu. The *Contour Options* dialog is shown in Figure 2-14 and contains three different tabs; Contours, Color Ramp, and Labels. The display options controlled under each tab are described in the following sections.

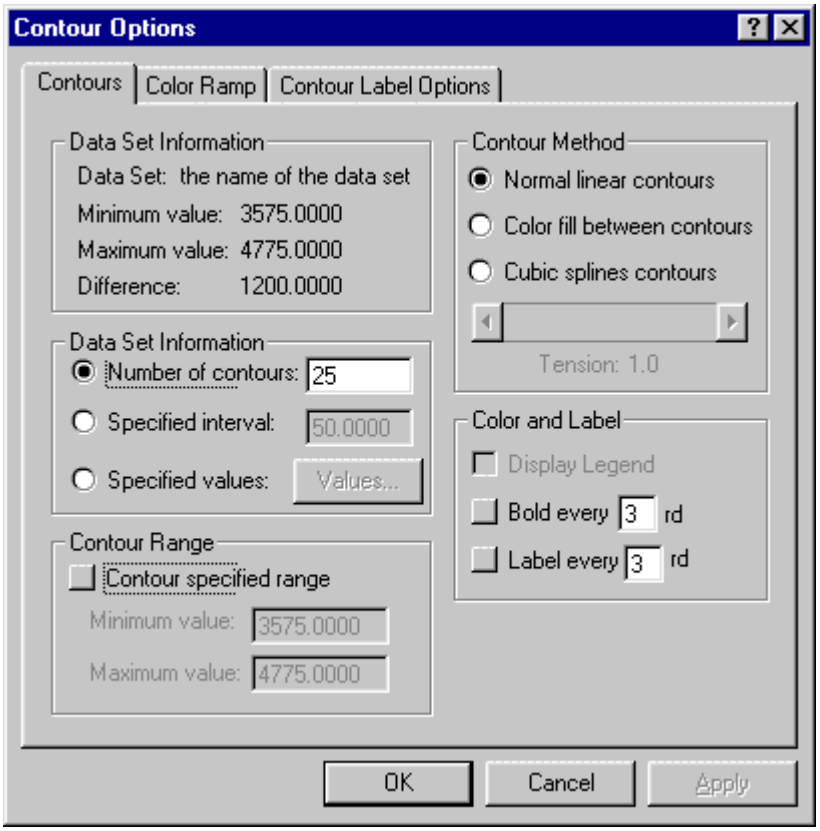


Figure 2-14 Contour Display Options Dialog.

Contours

The values shown in the upper left corner of the Contours tab (see Figure 2-14) correspond to the maximum and minimum values for the elevations of the active module (i.e., max and min DEM elevations when the DEM module is active). These values can be useful when choosing an appropriate contour interval.

The contour interval can be specified either by specifying a contour interval, a total number of contours (from which the contour interval is computed), or a set of explicit contour values. If the *Values* button is selected, a *Contour Values* dialog is displayed. Up to ten specific contour values can be typed into the dialog. A logarithmic scale can be set up in this way by specifying powers of ten as the set of values to be contoured. The *Set Log Scale* button in the *Values* dialog automatically defines powers of ten from .001 to 100.

Regardless of which of these options is selected, a maximum and a minimum contour value can be specified to restrict to the range of values contoured.

Contours at selected intervals can be automatically labeled and displayed with a thicker line width using the labeling options.

If the *Display Legend* option is selected, a vertical strip of colors with a legend of corresponding data set values is displayed in the upper left corner of the *Graphics Window* whenever the color ramp is used to display a TIN, DEM, etc.

The options at the upper right of the dialog control how the contours are computed. Three contouring methods are available:

1. The default method is *Normal Linear Contours* and causes the contours to be displayed as piece-wise linear strings.

2. If the *Color fill between contours* button is selected, the region between adjacent contour lines is filled with a solid color.

3. If the *Cubic Spline Contours* button is selected the contours are drawn as cubic splines. Drawing the contours as splines can cause the contours to appear smoother. Occasionally, loops appear in the splines or the splines cross neighboring contour splines. These problems can sometimes be fixed by adding tension to the splines.

Color Ramp

The Color Ramp tab (Figure 2-15) allows you to control the contouring color methods. There are three different coloring options:

1. If the *Ramp of Intensity* option is chosen, the ramp can be defined as a continuous (from white to through black) variation of the specified default color color.

2. If the *Ramp of Hues* option is chosen, the ramp can be defined as a continuous variation of hues (from blue through red) using the hue-saturation-value color model.

3. If the *Solid Color* option is chosen then all contours will be displayed using the specified default color.

For either of the first two options where color ramps are used you can specify the intensity/hue used for the minimum and maximum contour values. The ramp shown at the top of the *Color Scheme* group defines the available range of intensities/hues. The minimum intensity/hue corresponds to the location of the top slider bar and the maximum intensity/hue corresponds to the bottom slider. If the *Reverse* button is selected, the locations of the minimum and maximum intensities/hues change places. The larger ramp shown at the bottom of the *Color Scheme* group shows the currently defined range.

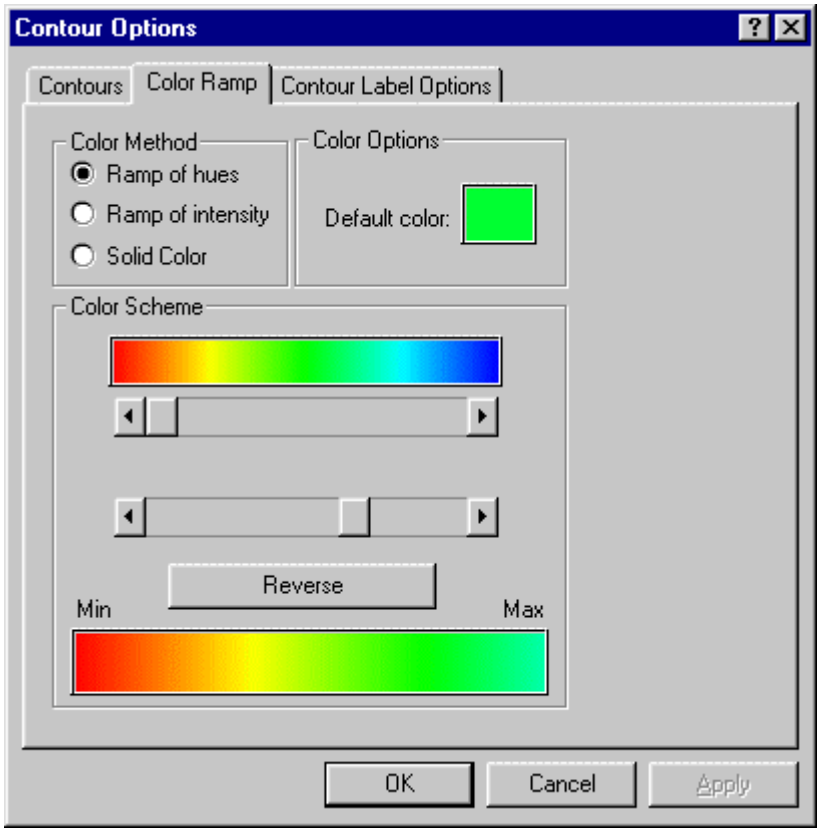


Figure 2-15. Color Ramp Options Dialog.

Labels

The *Labels* tab (Figure 2-16) in the *Contour Options* dialog is used to set the label color, label size, etc. of contour labels placed automatically or manually. The default spacing value controls the placement of labels when labels are generated automatically.

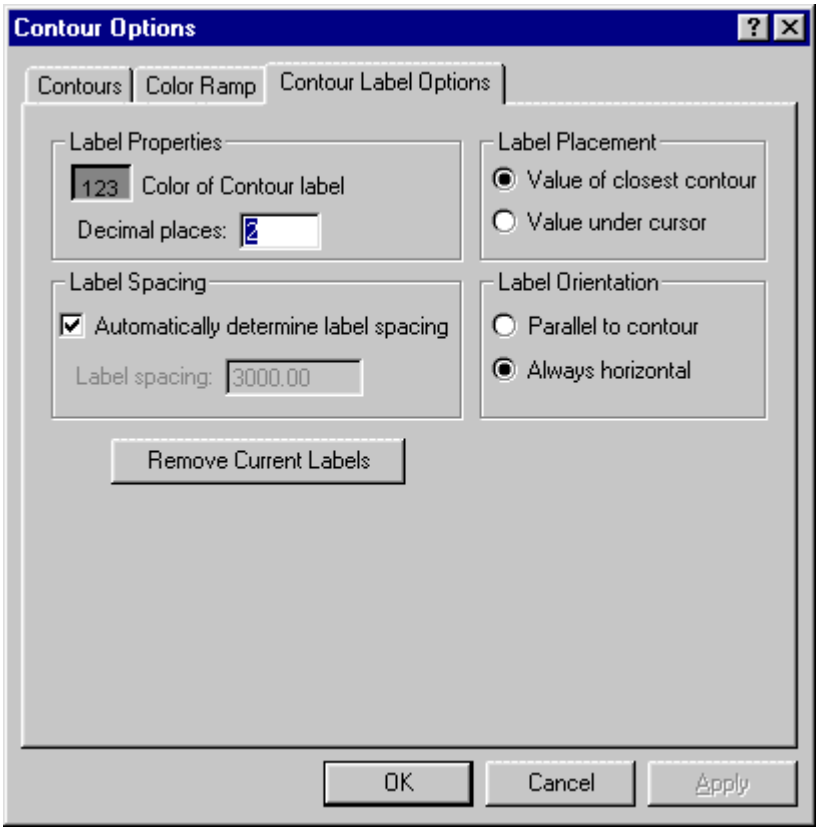


Figure 2-16 Contour Label Options Dialog.

Labels can be added to contours one of two ways:

1. If the contour label option is selected in the *Contours* tab, labels are automatically placed on the contours corresponding to the specified contour intervals.

2. In some modules, contour labels can be added manually to contours by selecting the *Contour Labels* tool in the *Tool Palette* and clicking on the contours where labels are desired. By default, the data set value corresponding to the point that was clicked is computed and a label corresponding to the nearest contour value is drawn centered at the point that was clicked. If the mouse button is held down, a box showing the outline of the label is drawn. The box can then be positioned precisely with the mouse. A line is drawn from the box to the point that was clicked to help the user keep track of the contour

that was selected. Contour labels can be deleted by holding down the *SHIFT* key while clicking on a label.

2.10.3 Refresh

When editing the image in the *Graphics Window* it occasionally becomes necessary to update the display or refresh the screen by redrawing the image. Whenever possible, *WMS* automatically updates the display. However, in several cases small parts may be obscured by editing procedures, and the display will need to be refreshed by selecting the *Refresh* command from the *Display* menu. The *Refresh* command has a macro in the tool palette .

NOTE: The process of redrawing can be aborted in many cases by pressing the *ESC* key.

2.10.4 Shading

The default display mode for a TIN or grid in the *Graphics Window* is a wire-frame image. Color shading and hidden surface removal can be applied to either TINs or grids in order to generate a realistic image. Hill shaded images of DEMs can also be created using the shade command with a DEM present.

Light Angle

The *Smooth Shading* option in the *Shading Options* dialog (brought up using the *Shading Algorithms* command in the *Display* menu) uses a light source defined by a lighting angle. The direction of the lighting angle can be set using the *Light Angle* command in the *Display* menu. This command brings up a dialog containing a diagram of a sphere. The sphere is shaded according to the current setting for the light angle. The light angle can be changed simply by clicking on the sphere with the cursor at the location where a direct ray from the light source to the center of the sphere would strike the sphere. The sphere shading is updated instantaneously to reflect the chosen light angle.

Shading Options

The *Shading Options* command in the *Display* menu can be used to change the technique used to shade the image in the *Graphics Window* when the *Shade* command is selected. The three available options are hidden edge removal, flat shading, and smooth shading. The options are only applicable when smoothing a

The *Hidden Line* option generates a wire-frame image with the hidden edges removed.

The *Flat Shade* option generates an image with the hidden edges removed but it also applies colors to the triangles of a TIN.

The *Smooth Shade* option is similar to the *Flat Shade* option except that a light source and Gouraud shading is used to make the TIN surface appear smoother.

Shade

The *Shade* command is used to generate a shaded image of a TIN or grid using the hidden surface removal technique, or a hill shade in plan view of a DEM in the *Graphics Window*. The default wire-frame image can be restored by selecting the *Refresh* command in the *Display* menu.

Images can be "draped" over a TIN if properly registered. This is controlled in the *Image Display Options* by choosing to map the image to the TIN. If an image is mapped it will be draped and shaded over the TIN when choosing the *Shade* command.

2.10.5 Drawing Grid Options

When entering new vertices or entering a polygon or polyline in plan view, it is often useful to have the coordinates snap to a uniform grid. This allows accurate placement of the objects when the desired coordinates are even multiples of some number.

A drawing grid can be activated using the *Drawing Grid Options* command in the *Display* menu. If the *Snap to grid* option is selected, all new vertices will snap to the closest grid point. The grid spacing and options for displaying the grid can also be set using the *Drawing Grid Options* dialog

2.10.6 Auto vs. Manual Redraw

When the *Manual Redraw* is on, all display refreshes must be done manually by either selecting the Refresh command in the Display menu or the *Refresh* macro from the tool palette. By default *WMS* is in *Auto ReDraw* mode which forces updates after display options change, or when a menu or dialog covers one of the graphics windows and there is not enough memory to save the pixels behind the menu/dialog so that they can be restored after the menu/dialog is dismissed. When in manual mode rather than forcing the update, the *Refresh* macro will turn red, signifying that the display is not up to date and refresh must be forced manually in order to bring it back up to date. The main advantage of using the manual redraw is when you are working on a computer that does not have enough video memory to do backing store and refreshes occur after each pull down menu disappears (typically only X-Window terminals have this problem).

2.10.7 Hide/Show Hydrograph Window

As described in section 2.2 the *Hydrograph Window* is a floating window and can be opened and closed. By default the *Hydrograph Window* will be closed to view. To open it you should select the *Show Hydrograph Window* command from the *Display* menu. The *Hydrograph Window* will appear as a floating window and can be closed by reselecting the command, which will have changed to *Hide Hydrograph Window*.

2.11 View Menu

The *View* menu is one of the standard menus in *WMS* and is available in all of the modules. The commands in the *View* menu are used to change the orientation of the image in the *Graphics Window*.

2.11.1 Frame Image

After altering the image display using the *Zoom* or *Pan* tools, the image can be centered by selecting the *Frame Image* command in the *View* menu. This command adjusts the window boundaries so that all currently visible objects just fit in the *Graphics Window*. It does not affect the viewing angle. The *Frame Image* command has a macro in the tool palette .

2.11.2 Set Window Bounds

The region of the real world coordinate system that is mapped to the *Graphics Window* can be altered using the *Pan* and *Zoom* tools. It is also possible to precisely control the visible region by selecting the *Set Window Bounds* command from the *View* menu. The *Set Window Bounds* dialog box appears and the x and y limits of the viewing area can be set.

2.11.3 Z Magnification

Occasionally an object may be very long and wide with respect to its overall depth (z dimension). It is possible to exaggerate the z scale so that the variation in the z value is more apparent by selecting *Z-Magnification* from the *View* menu and changing the magnification factor.

2.11.4 Changing the Viewing Angles

The objects in the *Graphics Window* can be rotated and viewed in three dimensions. Two angles, *bearing* and *dip*, are used to rotate the view. The bearing and dip values correspond to a rotation about the z and x axes. The bearing affects the horizontal angle (rotating the object in the xy plane), and

the dip changes the vertical angle (shifting the viewing angle on the object to a higher or lower perspective). The object cannot be tilted sideways. Using only two viewing angles rather than three is somewhat restrictive but it can be more intuitive.

Rotation

The viewing angles can be manipulated interactively with the *Rotate* tool as described earlier.

View Angle

The bearing and dip angles can be explicitly defined in the *Viewing Angle* dialog accessed by selecting the *View Angle* command from the *View* menu.

Plan View

Selecting the *Plan View* command changes the viewing angles so that the image is displayed such that the user is looking down the z-axis with the x-axis horizontal and the y-axis vertical. The *Plan View* command has a macro in the tool palette .

Oblique View

Selecting the *Oblique View* command restores the bearing and dip angles to their previously defined values. The Oblique View command has a macro in the tool palette .

View Last

Selecting the *View Last* command restores *Graphics Window* viewing parameters they were before the last viewing command was issued (rotate, zoom, pan, etc.).

CHAPTER 3

Map Module

3.1 Introduction

The Map module provides a suite of tools for defining watershed data in a GIS and then using the information to directly create a hydrologic model, or as a support utility for hydrologic data development with either TINs or DEMs. Other important tools of the map module include using drawing objects to enhance report documents, importing and converting DXF files, or mapping images to geo-reference data on the display. Tools in this chapter can be separated into four major components: Feature Objects, Images, DXF Files, and Drawing Objects.

Feature objects have been patterned after common GIS entities such as coverages (groups) of points, nodes, arcs (lines), and polygons, and can be created by importing *ArcView*® shapefiles (An *ARC/INFO*® coverage can be converted to a shapefile with the ARCSHAPE command in ARC).

Feature objects are used to establish a conceptual model of a watershed, i.e. arcs representing a stream network, polygons for basin boundaries and points or nodes for watershed and sub-basin outlets. Hydrologic models can then be created using any of the supported model interfaces.

A rough boundary and stream network can also be used to generate a TIN or two-dimensional finite difference grid that conforms precisely to streams and other important hydrologic features. Feature objects can also be used to create polygonal boundaries of soil type or land use to aid in the computation of curve numbers for hydrologic analysis.

Images can be used to provide a background display of a region, or draped over a TIN or grid as a texture map. They can be imported from TIFF files or created from within *WMS* by capturing the screen.

DXF files can be imported into and displayed with data created in *WMS*. DXF data can be used to create feature objects, or to enhance final project reports.

Drawing objects include text, lines, rectangles and ovals. These objects can be used for annotation and for providing more detail to displays created in *WMS* for project reports.

A section, with descriptions of the tools and commands, is given for each component of the map module.

3.2 Feature Objects

Feature objects in *WMS* have been patterned after Geographic Information Systems (GIS) objects and include points, nodes, arcs, and polygons. Feature objects can be grouped together into coverages, each coverage defining a particular set of information. The use of feature objects is determined by the coverage, or attribute set, to which they belong, but can be separated into three important categories: 1) As a means of defining basin polygons and stream networks of pre-delineated watersheds (typically this data would be imported as a shapefile from *ArcView*® or ArcInfo where the basin delineation and attribution has already taken place), 2) To define a conceptual model or layout of features in the watershed, such as its rough boundaries and streams. This conceptual model is then used to aid in the construction of a TIN or DEM processing for delineating watershed and sub-basin boundaries, and 3) soil, land use, rainfall, or other data which can be used to define important hydrologic modeling parameters such as curve number (CN) or rainfall zone.

3.2.1 Creating Watershed Models Directly

With an ever-increasing availability of GIS and other digital data, delineated stream networks and basin boundaries for a given watershed may already exist. In order to take advantage of this type data when available, *WMS* allows hydrologic models to be built directly from three different features of the map module: polygons representing basin boundaries, arcs representing a stream network, and nodes representing watershed and sub-basin outlet points.

This means that data imported from an *ArcView*® shapefile can be used directly to set up the hydrologic model. Further, since attributes from the shapefiles can also be read in, much of the hydrologic data developed with the GIS tool can be used to define input parameters of the given hydrologic model. It also means that a tiff image map or other data can be used to establish the boundaries of the watershed at the proper scale so that lengths and areas

determined from the feature objects are correct, or simply used as a scaled schematic representation of the watershed (in such cases area and length values would have to be determined by some other means and defined in appropriate dialogs prior to running one of the supported hydrologic models).

3.2.2 Creating TINs or Finite Difference Grids

The initial versions of *WMS* were based on watershed delineation from TINs. However, many hours of edge swapping and breakline insertion was required to develop a TIN that was suitable for basin delineation. This was because triangulation of an arbitrary set of points does not guarantee that triangle edges will correspond to drainage features such as streams and ridges. But, if a set of pre-defined streams, ridges and boundary arcs are used to guide the TIN construction, triangle edges can automatically be created to conform to these features, and at the same time inherit any associated attributes. In such cases, either a DEM or an existing TIN can be used as a background elevation map for interpolating z values to newly created vertices of the TIN. The *Create TIN* command, discussed later in this chapter, details this process.

The same process can be used to create finite difference grids used by the CASC2D model. For this model, attributes assigned to the points, arcs, and polygons are used to map model parameters to the resulting grid.

3.2.3 Mapping Curve Numbers and Other Analysis Parameters

Besides the creation of stream networks, and sub-basin boundaries, feature objects can be used to define polygonal zones representing soil types, land use, rainfall zones, etc. These polygons can then be overlaid with the basin boundaries to determine composite curve numbers, predominant soil type, and other parameters required by the supported hydrologic models. (see section 15.2)

3.2.4 Feature Object Types

The definition of feature objects follows the paradigm used by typical GIS software that supports vector data. Basic object types include points, nodes, vertices, arcs, and polygons. The relationship between these objects is illustrated in Figure 3-1. By patterning the *WMS* data structures in this way it easy to share files with GIS databases such as *ARC/INFO*®.

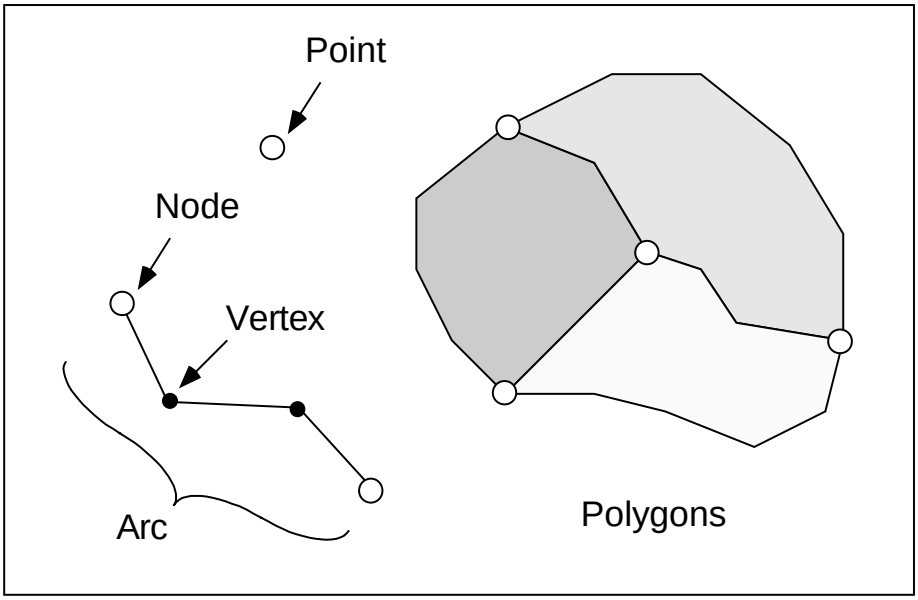


Figure 3-1 Feature Object Types.

Points

Points are xy locations that are not attached to an arc. Points have unique id numbers and attributes. Points are used in *WMS* when importing a set of xy (or xyz values) for the purpose of creating arcs or polygons.

Nodes

Nodes define the beginning and ending xy locations of an arc. They typically occur at branching points or other places where boundary conditions need to be assigned. Nodes have unique ids and attributes.

Vertices

Vertices are xy locations along arcs in between the beginning and ending nodes. They are used to define the geometry of the arcs and do not have ids or attributes.

Arcs (Lines)

Arcs, or lines, are a series of connected vertices with a beginning and ending node. Arcs may have any number of vertices between nodes. Whenever a point is used in the creation of an arc it is automatically converted to a node. Arcs are used to represent streams, or ridges, and are used in the creation of polygons.

Polygons

Polygons are a group of connected arcs that form a closed loop. Polygons may not overlap within a single coverage. However, a polygon can have a hole defined by having a set of closed arcs defining interior polygons. An example of such a case is shown in Figure 3-2 where three arcs are used to define two polygons. Polygon A is made up of arcs 1, 2 and 3, whereas polygon B is defined by a single arc (number 3). For polygon A arcs 1, 3 and 4 define the exterior boundary whereas arc 2 defines a hole.

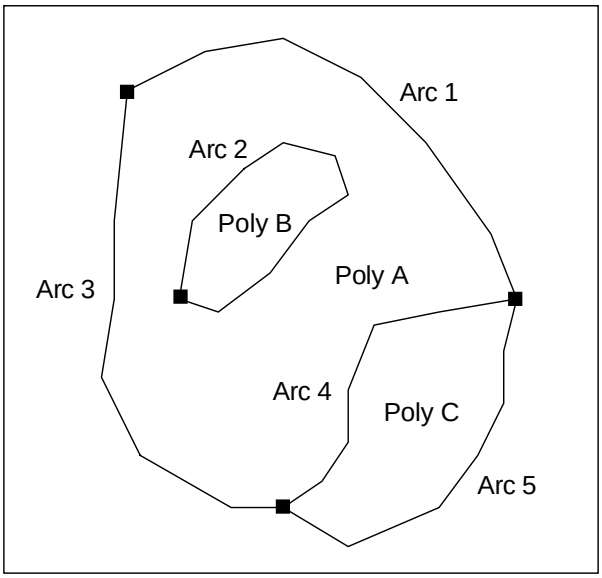


Figure 3-2 Polygon With Holes.

Coverages

Feature objects can be grouped together into coverages. Each coverage represents a particular set of data. For example, one coverage can be used to define land use, and another coverage can be used to define soil type. Coverages are described in more detail later in this chapter.

A common use for coverages is defining soil type and land use for Curve Number (CN) computation from polygons. Separate coverages must be used for the land use and soil type polygons, since polygons may not overlap within a given coverage.

Coverages are managed using the *Coverages* dialog accessible from the *Feature Objects* menu shown in Figure 3-3.

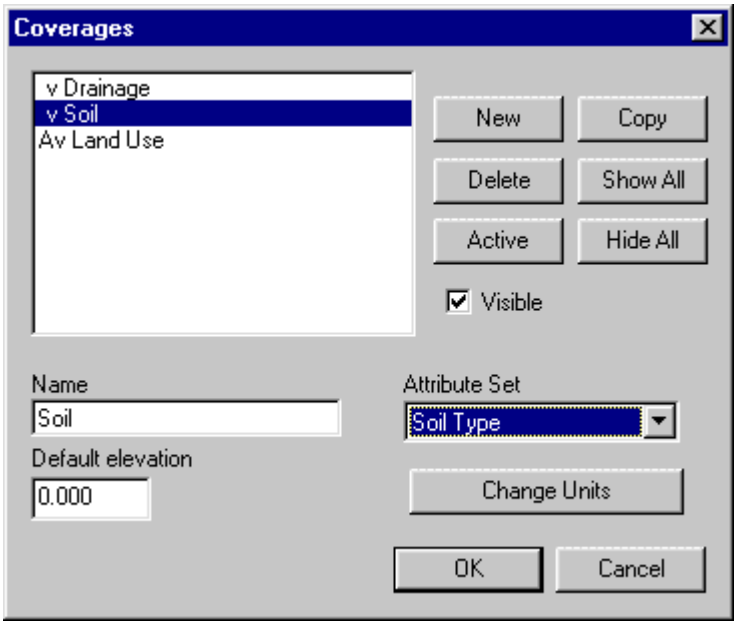


Figure 3-3 Coverages Dialog.

Since multiple coverages may exist for the same model, one must be designated as “active”. When new objects are created they are placed in the active coverage. Furthermore, the active coverage is the only coverage which can be edited (i.e. arcs, points, nodes, etc. selected and modified).

A coverage can be activated by double-clicking on the name or selecting the name and then the *Active* button. Each time *WMS* is started a default coverage is created, but new coverages can be created using the *New* button. When a new coverage is created it automatically becomes the active coverage. An entire coverage can be deleted by selecting the coverage name and then the *Delete* button.

Since you may need to change the active coverage frequently and from different modules within *WMS* a combo-box listing the currently defined coverages can be accessed from the edit window (the space in the GUI just below the menus). You can change the active coverage by selecting it from the drop-down combo box without having to access the *Coverages* dialog.

A coverage can be either visible or hidden. Only the visible coverages are displayed when drawing to the *Graphics Window*. When the coverage is visible a small "v" is placed to the left of the coverage name in the *Coverages* dialog. The visibility status can be set in one of two ways:

1. Clicking on the coverage name and setting the *Visible* toggle to the desired status.

2. The Show All and Hide All buttons can be used to make all coverages either visible or hidden.

Each coverage has an associated attribute set. The attribute set determines which types of attributes can be associated with the points, nodes, arcs, and polygons of the coverage. The different attribute sets are discussed in the next section.

The coverage name is changed by first selecting the coverage and then changing the name in the name edit field. The default elevation is used when displaying the coverage in an oblique view. All objects in a coverage are assigned the same elevation.

3.2.5 Feature Object Tools

The following feature object tools are in the dynamic portion of the *Tool Palette* when the *Map* module is activated. These tools relate only to the feature objects. The remaining tools in the dynamic portion of the tool palette are discussed in the section on Drawing Objects.

Only one tool is active at any given time. The action that takes place when you click in the *Graphics Window* with the cursor depends on the current tool. Tools are used to either create feature objects, define attributes for a selected object using the type dialogs described in the previous section or operate on selected objects using one of the commands described in a later section.

Select Points/Nodes

The *Select Points/Nodes* tool is used to select existing points or nodes. A selected point/node can be deleted, moved to a new location, or operated on by one of the commands in the *Feature Objects* menu. The coordinates of selected points/nodes can also be edited using the *Edit Window*. Double clicking on a point or node with this tool brings up the *Point* or *Node Attribute* dialog.

Select Vertex

The *Select Arc Vertices* tool is used to select vertices on an arc. Once selected, a vertex can be deleted, moved to a new location, or operated on by one of the commands in the *Feature Objects* menu. The coordinates of selected vertex can also be edited using the *Edit Window*

Select Arc

The *Select Arc* tool is used to select arcs for operations such as deletion, redistribution of vertices, or building polygons. Double clicking on an arc with this tool brings up the *Arc Attribute* dialog.

Create Points

The *Create Points* tool is used to interactively create new points using the cursor. A new point is created for each location the cursor is clicked on in the *Graphics Window*. A background drawing grid can be turned on using the *Grid Options* command in the *Display* menu to aid in the placement of points.

Create Arc Vertex

The *Create Arc Vertex* tool is used to interactively create new vertices along an existing arc, to add more detail. A new vertex is created for each location the cursor is clicked on in the *Graphics Window*, that it is within 10 screen pixels of an existing arc. Once the vertex is created, it can be repositioned with the *Select Vertex* tool.

Create Arc

The *Create Arc* tool is used to interactively create new arcs. An arc is created by clicking once on the location where the arc is to begin, clicking once to define the location of each of the vertices in the interior of the arc, and double-clicking at the location of the end node of the arc.

As arcs are created, it is often necessary for the beginning or ending node of the arc to coincide with an existing node. If you click on an existing node (within a given pixel tolerance) when beginning or ending an arc, that node is used to define the arc node as opposed to creating a new node. Also, if you click on a vertex of another arc while creating an arc, that vertex is converted to a node and the node is used in the new arc. If an existing point is clicked on while creating an arc, the point is converted to a vertex, unless it is the beginning or ending location of an arc, in which case it is converted to a node.

While creating an arc, it is not uncommon to make a mistake by clicking on the wrong location. In such cases, hitting the *BACKSPACE* key backs up the arc by one vertex. The *ESCAPE* key can also be used to abort the entire arc creation process at any time.

The new arc type is determined from the *Feature Arc Type* dialog, accessed from the *Attributes* command in the *Feature Objects* menu. The *Feature Object Type* dialog that comes up when selecting the *Attributes* command is determined by the currently active tool. For example if the *Select Arc* or *Create Arc* tools are active the *Feature Arc Type* dialog comes up, whereas if the *Select Polygon* tool is active the *Feature Polygon Type* dialog comes up.

When creating stream arcs, the points/vertices must be connected from downstream to upstream as the arc is entered. Stream arcs are drawn with an

arrow pointing in the downstream direction and can be reversed after creation using the *Reverse Directions* command.

Select Polygon

The *Select Polygon* tool is used to select previously created polygons for operations such as deletion, assigning attributes, etc. A polygon is selected by clicking anywhere in the interior of the polygon. Double-clicking on a polygon with this tool brings up the *Polygon Attributes* dialog.

Select Branch

The *Select Branch* tool can be used to automatically select all arcs of a branch without having to select each individually. This is particularly useful when locally redistributing vertices along a stream branch. A branch is selected by selecting any arc in the branch. *WMS* searches down the stream from the selected arc until the next branching node is encountered, and then adds all arcs upstream from that node to the list of selected arcs.

Select Network

The *Select Network* tool can be used to select all arcs of a given network. It works similarly to the *Select Branch* tool except that when an arc is selected *WMS* searches downstream until a node with no downstream arcs is encountered instead of stopping at the next downstream branching node.

3.2.6 Coverage Type

Currently *WMS* supports the *HEC-1*, *TR-20*, *TR-55, NFF, MODRAT, Rational Method,* and *CASC2D* hydrologic models, and in the future other models will be supported. Since each model requires different input options, the capability to define the model specific options for feature object data must exist. In *WMS* this is accomplished by setting a type for each coverage that corresponds to the type of analysis being performed. There are five different options for coverage types: drainage, land use, soil type, CASC2D, runoff coefficient, time computation, Rainfall Zones, DPA Zones, TC, Xmodel, cross section, and general.

Attributes for the different feature objects are defined in one of two ways:

1. By selecting the feature object (point, node, arc, polygon) and then choosing the *Attributes* command from the *Feature Objects* menu.

2. By double-clicking on the feature object.

The dialog that appears depends on the currently selected object and the active coverage. Remember that the only feature objects which can be selected/edited are the feature objects of the active coverage. The active coverage can be changed from the edit window using the drop-down combo box or from the Coverages dialog shown in Figure 3-3. The feature object attributes (and dialogs used to set them) for the different coverage types are explained in the following sections.

Drainage Coverage

The Drainage Coverage is the primary coverage used by *WMS*. When ever you start a new *WMS* session an "empty" coverage is created and assigned the drainage coverage type. If you begin creating points, arcs, and polygons, by default they will belong to this drainage coverage. Most of the work you do will be centered around the drainage coverage which has two different purposes as outlined in the following paragraphs:

1. To develop a hydrologic model directly from feature objects, or GIS vector data. In version 5.0 and higher of *WMS*, the points, arcs, and polygons in the drainage coverage are tied directly to the hydrologic modeling tree. When a stream arc is created, the most downstream node of the stream arc is converted to an outlet node and an outlet is added on the hydrologic modeling tree. A drainage basin is also added on the tree for every upstream arc from an outlet node. If a drainage polygon is created and a stream is located in the boundaries of that drainage polygon, the drainage polygon is tied into the tree as a drainage basin. An example of a drainage coverage used for the purpose of creating a hydrologic model is shown in Figure 3-4.

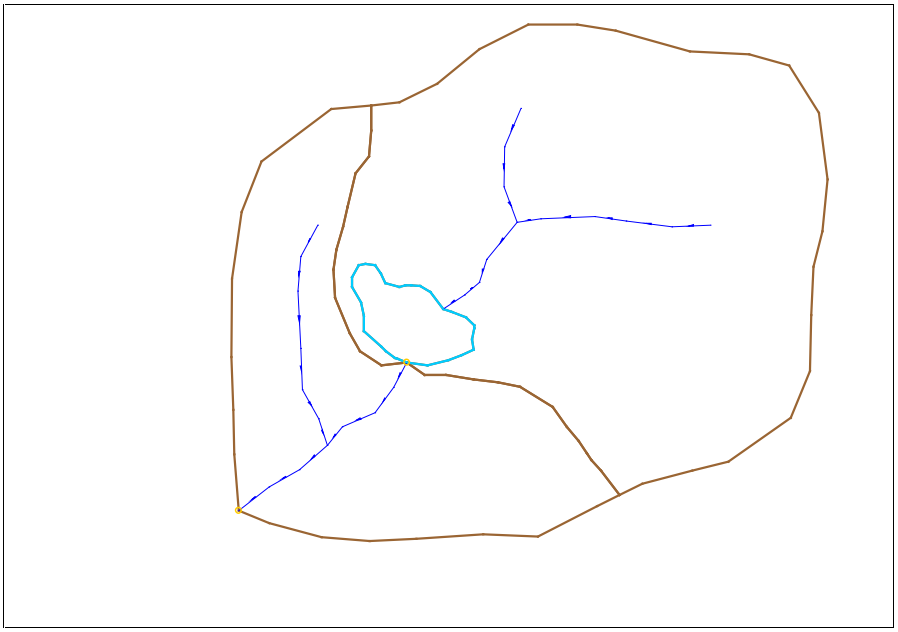


Figure 3-4 Example Drainage Coverage Used to Create a Hydrologic Model.

2. To use as a "conceptual" model when creating a TIN from a background elevation data source and feature objects for the purpose of automated watershed delineation. *WMS* can use TINs for performing watershed delineation. However, when triangulating a set of scattered xyz data, it is difficult if not impossible to ensure that triangle edges conform to streams, ridges and other drainage features. Drainage coverage feature objects can be used to create a TIN that conforms to the feature objects. The distribution of TIN vertices will approximate the distribution of feature arc vertices in the conceptual model.

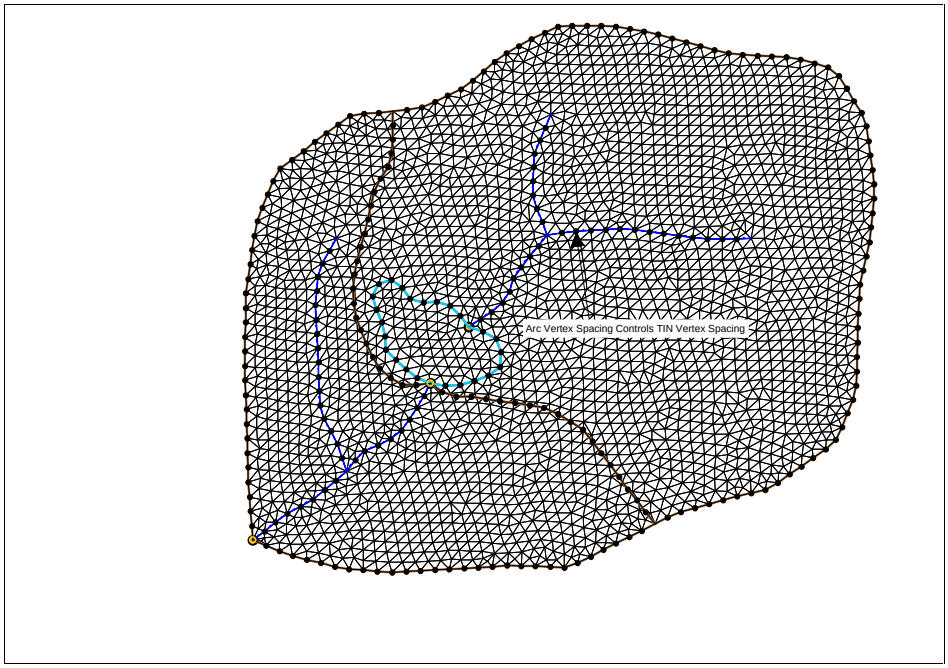


Figure 3-5 Example Drainage Coverage Used as a Conceptual Model for Creating a TIN.

In either case the same set of attributes can be defined, but are used for a different purpose. The following sections outline the different attributes available for points, arcs, and polygons in the drainage coverage and describe the differences when used to create hydrologic models or TINs.

Drainage Point/Node Types

Drainage points can have either generic or outlet attributes. Generic points have no attributes and can be used for drawing purposes. Outlet points are directly linked to the tree. By default, the most downstream node of a stream arc is an outlet point and its attributes cannot be changed. If additional points along a stream are converted to outlet points, the hydrologic modeling tree is updated when these outlet points are added. Furthermore, if outlet points are converted back to generic points, the hydrologic modeling tree is also updated.

When creating a TIN from feature objects, outlet points on the feature objects are converted to outlet points on the TIN. If points are imported but not used by any arcs they can be removed using the *Delete Isolated Points* command.

Drainage Arc Types

The *Arc Type* dialog used for drainage coverages is shown in Figure 3-6. *Generic* arcs have no attributes and are used when developing drainage boundaries or when establishing the boundary polygon for creating a TIN from feature objects.

Figure 3-6 Feature Arc Type Dialog for Drainage Coverages.

Stream arcs should be used to define stream reaches hydrologic models. The direction of a stream arc is critical and in *WMS*, the direction of flow is opposite the direction the arc is created. In other words you should create stream arcs from downstream to upstream (the "from" node being the downstream node and the "to" node being the upstream node). If an arc is designated as a stream, then an outlet point is added at the most downstream node of the stream arc, and the hydrologic modeling tree is updated to include this outlet point. When a stream arc is created, A drainage basin is added on the hydrologic modeling tree for each upstream arc emanating from an outlet point. If the stream arcs are being used to create a TIN, a breakline is forced along the arc (triangle edges are enforced along the arc). Furthermore, a stream along the affected triangle edges is automatically created for the TIN.

Lake arcs should be used to trace around the boundary of a lake. *Stream* arcs can be attached up and/or downstream of the lake. If no downstream stream arc is attached to the lake then a node on the lake will automatically be defined as the outlet of the watershed and a drainage basin will be created for the lake/stream combination. If a downstream arc is defined from the lake, then the downstream most point on the stream will, by default, be designated as the watershed outlet. However, any node on a lake arc can be assigned an outlet if you want to define a sub-basin outlet at the lake. If a lake polygon exists when using feature objects to create a TIN, then the lake arcs will be enforced in the TIN as breaklines.

Ridge arcs should only be used when creating a TIN from feature objects and are used to designate any other (besides boundary, stream, and lake) linear segment you wish to have enforced as a breakline in the resulting TIN. They have no effect when creating the hydrologic model directly from the feature objects.

Drainage Polygon Types

The *Polygon Type* dialog used for drainage coverages is shown in Figure 3-7.

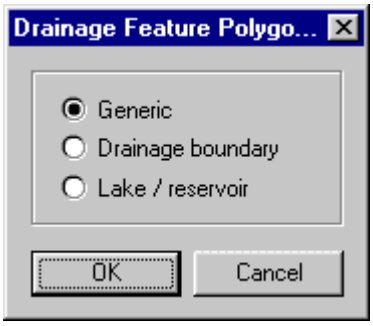


Figure 3-7 Feature Polygon Type Dialog for Drainage Coverages.

To link a polygon to the hydrologic modeling tree in the map module, the polygon must be a *Drainage boundary* polygon type. Also, the polygon must have a stream inside the polygon. Each stream has a drainage basin assigned to it. This drainage basin will also be assigned to the polygon containing the stream, provided the polygon is a Drainage boundary polygon. Because most of the time you will be working want to use drainage polygons *WMS* uses the following defaults when building polygons in a drainage coverage:

1. If the arcs that enclose a polygonal region contain at least one generic arc type the resulting polygon will be assigned a drainage boundary type.

2. If all the arcs that enclose a polygonal region are lake arcs then the resulting polygon will be a lake polygon.

The generic polygon type is really only used as an intermediate polygon type when importing data from another source.

When creating a TIN, it is only necessary to have a single bounding polygon (of either *Generic* or *Drainage boundary* type) to define the extent of the area that will be converted to a TIN. If you have multiple sub-basins, *WMS* will ignore sub-basin boundary arcs interior to the watershed (you must define the sub-basins boundaries from the TIN).

Land Use Coverage

A Land Use coverage in *WMS* has a different purpose depending on the model and/or model parameters you wish to assign to basins using land use as an indicator. The following is a list of operations that can be done using land use coverages, and the parameters that must be linked to land use ID's:

1. SCS (NRCS) Curve Numbers for hydrologic soil groups (requires the use of a soil coverage as well).

   Soil type A CN, soil type B CN, soil type C CN, and soil type D CN.

2. Green & Ampt parameters for infiltration modeling in *HEC-1*.

Initial abstraction, percent impervious, percent vegetation cover.

3. HSPF Pervious and Impervious land segments and other parameters.

You indicate which application(s) you wish to use the land use coverage for by selecting the appropriate toggle box in the upper right portion of the *Land Use Mapping* dialog (see Figure 3-8). You may only define parameter values for the applications selected.

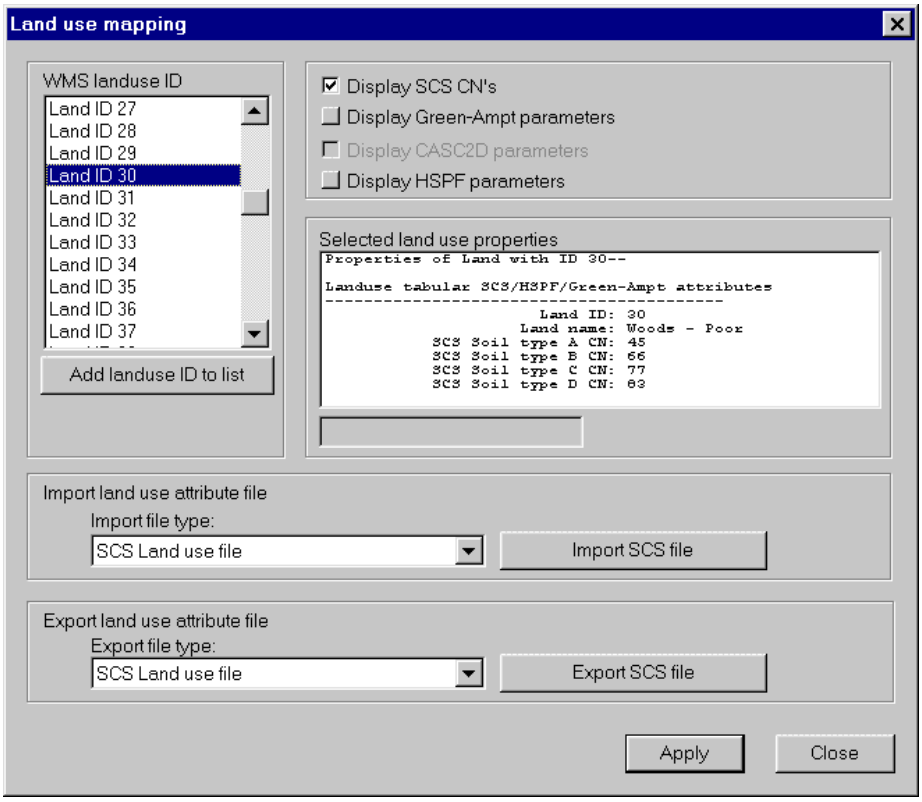


Figure 3-8 Land Use Mapping Dialog.

Each land use polygon will have a land use ID associated with it (a single integer number). In order to perform the correct mapping you will need to link the appropriate land use variables to each land use ID. *WMS* allows you to do this in one of two ways. First of all, with the land use coverage active, you can open the *Land Use Mapping* dialog from the *Feature Objects* menu using the *Attributes* command, and then create new IDs and enter parameters for each ID. Secondly, you can enter the data in a text file and then import it from within the same dialog. The file format is simple and is defined in section 20.17.

Parameter values for land use IDs are defined by selecting the ID in the *WMS landuse ID* text window and the parameter from the *Selected landuse properties* text window, and then entering the value in the edit field.

If you choose to define all the data manually using the *Land Use Mapping* dialog, you will want to export the data to a file so that you will not need to reenter the data for future models using the same land use parameter definitions.

Once the land use parameters have been defined, land use ID's can be assigned to polygons by first selecting the desired polygon(s) and then choosing the *Attributes* command from the *Feature Object* menu (or by double-clicking on the polygon) and choosing the appropriate ID from the *WMS* landuse ID text window.

Mapping files may also be imported/exported so that if you constantly use the same landuse id attributes you do not need to enter the values for each new model.

Once land use IDs have been assigned to polygons and parameters linked to the land use IDs, model parameters can be computed using the *Compute Composite Runoff Coeff/CN…* command from the *Calculators* menu in the *Hydrologic Modeling* module (see section 15.2).

Soil Type Coverage

The soil type coverage is similar to the land use coverage in that it can be used to map different model parameters (related to soil type) from polygonal coverages (usually imported from a GIS). The following is a list of operations that can be done using soil type coverages and the parameters that must be linked to soil ID's:

1. Hydrologic soil group to map SCS Curve Numbers (requires the use of a land use coverage as well).

   Soil Type Number (must be = 0 for type A, 1 for type B, 2 for type C and 3 for type D).

2. Runoff coefficients for the Rational Method (C in Q = CiA).

   Runoff coefficient.

3. Green & Ampt parameters for infiltration modeling in *HEC-1*.

   Hydraulic conductivity, percent impervious, percent effective.

You indicate which application(s) you wish to use the soil type coverage for by selecting the appropriate toggle box in the upper right portion of the *Soil Type Mapping* dialog (see Figure 3-9). You may only define parameter values for the applications selected.

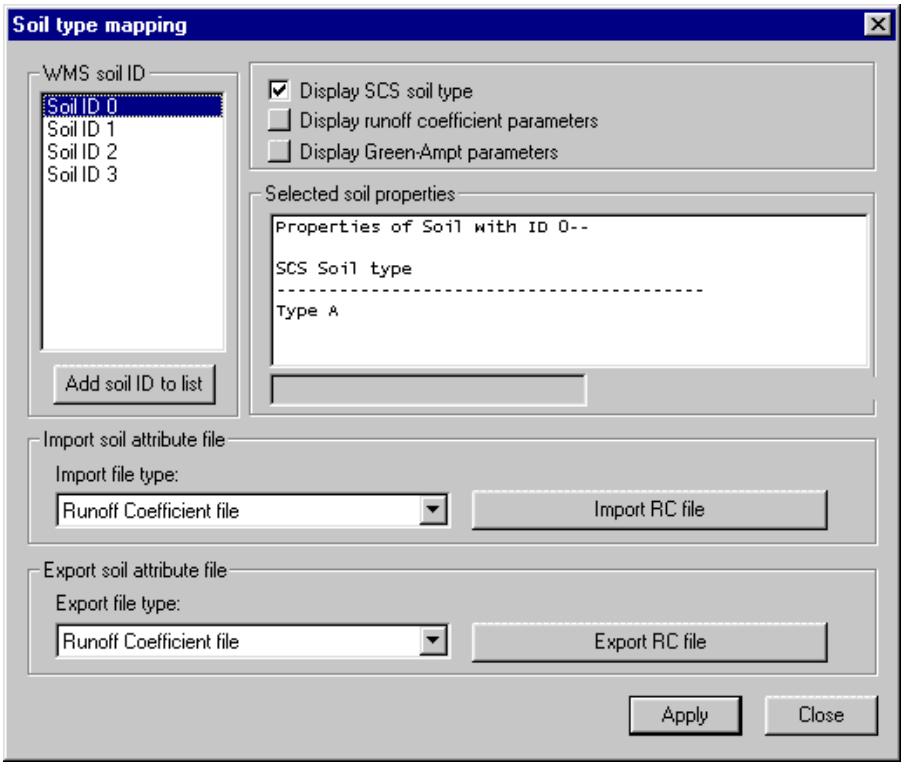


Figure 3-9 Soil Group Dialog.

Each soil type polygon will have a soil type ID associated with it (a single integer number). In order to perform the correct mapping you will need to link the appropriate soil type variables to each ID. *WMS* allows you to do this in one of two ways. First of all, with the soil type coverage active, you can open the *Soil Type Mapping* dialog from the *Feature Objects* menu using the *Attributes* command, and then create new IDs and enter parameters for each ID. Secondly, you can enter the data in a text file and then import it from within the same dialog. The file format is simple and is defined in section 20.18

Parameter values for soil type IDs are defined by selecting the ID in the *WMS soil type ID* text window and the parameter from the *Selected soil type properties* text window, and then entering the value in the edit field.

If you choose to define all the data manually using the *Soil Type Mapping* dialog, you will want to export the data to a file so that you will not need to reenter the data for future models using the same soil type parameter definitions.

Once the soil type parameters have been defined, soil type ID's can be assigned to polygons by first selecting the desired polygon(s) and then choosing the *Attributes* command from the *Feature Object* menu (or by double-clicking on the polygon) and choosing the appropriate ID from the *WMS* soil ID text window.

Once soil type IDs have been assigned to polygons and parameters linked to the soil type IDs, model parameters can be computed using the *Compute Composite Runoff Coeff/CN…* command from the *Calculators* menu in the *Hydrologic Modeling* module (see section 15.2).

Runoff Coefficient Coverage

The Runoff Coefficient coverage is used in the same was as the land use coverage, except that rather than defining a land use type a floating point runoff coefficient can be entered for each polygon. Composite runoff coefficients used in rational method simulations can then be computed using the *Composite Runoff Coeff/CN…* command in the *Calculators* menu of the *Hydrologic Modeling* module.

Time Computation Coverage

The *Time Computation* coverage type allows you to define equations along arcs for computing flow path travel times within a basin. In this way times of concentration or lag times can be determined using standard overland, and channel flow equations such as those used by the FHWA, *TR-55*, or any other user-defined equation. Since most equations used for travel time are functions of flow path length and slope, *WMS* automatically determines the length of the arc, and if a TIN or DEM is available the slope, and makes them available for use in an equation.

Time of concentration arcs are similar to stream arcs as defined in the drainage coverage in that their direction is important. When creating time computation arcs you should always define them from “downstream” to “upstream” in the same way that drainage arcs are defined.

Time computation arcs are the only type of arcs in the *Time Computation* coverage. Each arc is assigned a time of travel equation. Equations may be selected from a library of equations or as a user-defined equation. The predefined equations in *WMS* include TR55, FHWA, and Maricopa County. For example, TR55 includes the standard equations for sheet flow, shallow concentrated flow, and open channel flow. The dialog shown in Figure 3-10 is used to assign the arc attributes and corresponding values for the parameters of the equation so that a travel time for the arc can be determined.

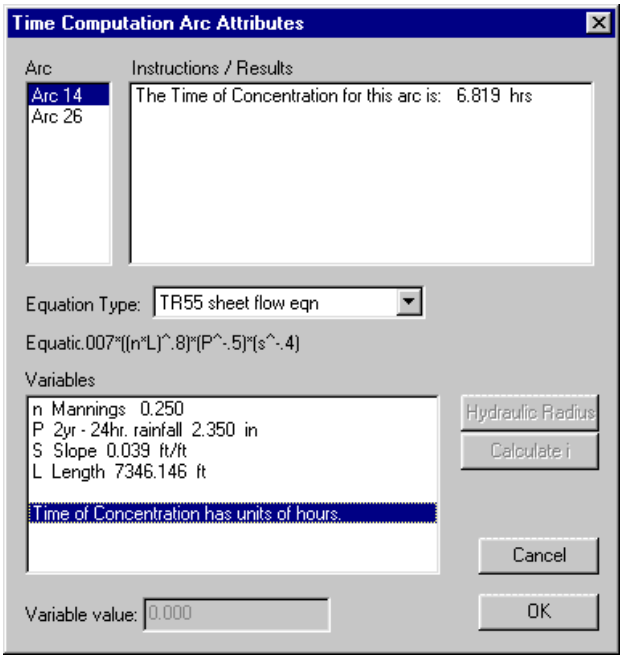


Figure 3-10 Time Computation Arc Attributes Dialog.

Time of concentration or lag time for a basin is determined by summing the travel times of all time computation arcs within a basin. A summary of how this is automated within *WMS* and further information on defining equations and computing travel times for time computation arcs is given in the hydrologic calculators chapter in section 15.3.

There are no point/node or polygon attributes in *Time Computation* coverages.

MODRAT Coverages

F0601 is a modified rational method program developed by Los Angeles County and is only useful for hydrologic analysis in LA County. There are three different coverage types that have been implemented in *WMS* to aid in the mapping of important F0601 parameters from GIS coverages. These coverages include rainfall zone, debris production zones (DPA zones), and time of concentration data. Further, variations within the soil type and land use zones are also used with F0601 data. Use of these coverage types is described in a separate manual delivered to users of the F0601 model.

Cross Section Coverage

The *Cross Section* coverage type can be used to view the cross section of any polyline on a TIN or DEM. *WMS* does not include an interface to a hydraulic model such as *WSPRO* or *HEC-RAS*, but cross sections can be used to compute normal depth for a given flowrate (or flowrate for a given depth) in the channel calculator (see section 15.4). They can also be exported into a space-delimited file and then viewed in a spreadsheet application or copied

into a hydraulic model file. While the primary use of the cross section coverage is to develop cross sections, they can be used to view the profile of any polyline (i.e. stream, ridge, etc.) on a TIN or DEM.

Cross section coverages are made up of only arcs (nodal and polygonal attributes cannot be defined). However, the placement of vertices along the arc is important since *WMS* will interpolate an elevation for the cross section at each vertex. If you want to add more detail to the cross section after it has been created you can use the add vertex tool to place new vertices on the cross-section at locations that you wish to interpolate an elevation.

Once the cross section is created you can view the station-elevation information along with a profile plot in the *Cross Section Editor* (see Figure 3-11) by selecting the arc and choosing the *Attributes* command from the *Feature Objects* menu (or by double-clicking on the arc when the select arc tool is active).

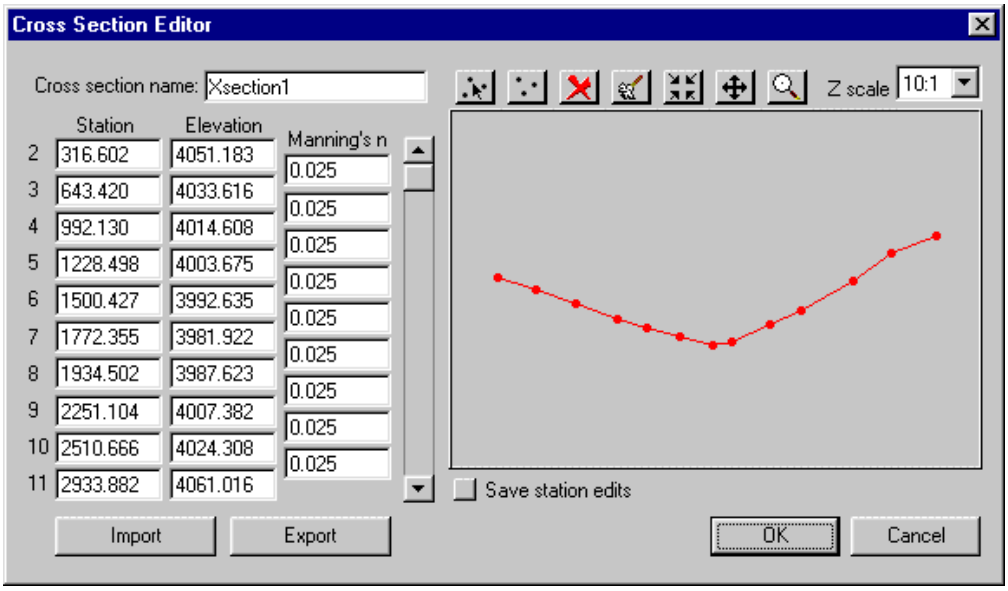


Figure 3-11 Cross Section Editor Dialog.

The *Cross Section Editor* shows the station and elevation of each of the vertices on the arc. The station values are determined by starting from 0.0 and then computing the distance between each vertex from their xy coordinates. Elevations are determined by interpolating from either a TIN or DEM. If both a TIN and a DEM are present then the TIN will take precedence over the DEM. Manning's n values can be defined for each segment of the cross section and are used when computing normal depth in the channel calculator (see section 15.4). The interpolated elevations can always be edited either by changing the value in the appropriate edit field or by selecting the point and dragging up and down in the plot window. Stations can only be edited if the *Save station edits* is toggled on. However, once you start editing the stations

they it will remove the detail in the xy plane since a cross section only has x and z values. In other words the stations will appear as a straight line on the DEM (the elevations will remain the same as when they were originally interpolated and stations not edited will remain unaffected). In general it is better to edit stations by adding/deleting/moving arc vertices directly rather than trying to edit in the *Cross Section Editor*. The several tools are used to select cross section points , create new ones , delete selected ones , or control the display by refreshing , framing , panning , or zooming . Since it is often difficult to visualize the cross section when the z scale is the same as the x (station), you can change the z scale to something more appropriate using the drop-down combo box. Finally, the name of the cross section is important because if you want to compute hydraulic properties for it from the cross section calculator you will need to be able to identify it. By default *WMS* will name a new cross section arc XsectionID (where ID is the ID of the arc), but you may wish to change it to be something more identifiable.

The *Export* button allows you to save the current cross section to a space-delimited file (station elevation n) that can be imported in a spreadsheet or hydraulic modeling program. The *Import* button allows you to import a space-delimited file.

If you wish to analyze a surveyed cross section or one that is not cut from a TIN or DEM in *WMS*, you can enter the station-elevation information directly. To do so you will first need to create an arc (don't worry about having intermediate vertices though), then in the *Cross Section Editor* you can edit the end station and elevations and insert new points on the cross section for each of the points along the profile (be sure to turn on the *Save station edits* toggle). *WMS* will create arc vertices for each of the interior points, but will use the elevation values you have provided rather than interpolating from a TIN or DEM.

Rain Gage Coverage

The rain gage coverage was designed specifically for use with the HEC-1 interface in *WMS*. The HEC-1 interface allows the x-y coordinates of gages to be defined (see section 10.14) and then a series of thiessen polygons created from the gages. When gages are present, weights are assigned to each basin from the thiessen polygon network as the basin data are computed. Using the feature point create/select tools gages can be created/edited in a rain gage coverage by graphical selection as opposed to the HEC-1 interface where the coordinates are typed in explicitly. As feature points are created/edited a rain gage is simultaneously created (note that the feature point and the rain gage point are separate because it is still possible to create the gages without a rain gage coverage by typing in the coordinates in the HEC-1 interface). The only feature in the rain gage coverage is points. Rain gage display options are

found under the *Drainage* tab (if you are asking why it is clearly a good question, but "historical reasons" is the only excuse we can provide).

General Coverage

Sometimes you may be unsure which coverage attributes to use, especially when importing a layer from another source (GIS, DXF). The general coverage allows you to create/edit points/nodes, arcs, and polygons without defining attributes, or worrying about special rules associated the different coverages and attributes. For examples *WMS* requires that stream and time computation arcs be defined from downstream to upstream. If you import data from another source you have no guarantee of the ordering, and if you import them to a Drainage Coverage type it may cause problems. Therefore, you can import them to a general coverage, make appropriate edits (streams can be reordered by selecting the downstream most point and choosing the *Reorder Streams* command from the *Feature Objects* menu), and then convert the coverage to a drainage type.

XMODEL Coverage

The XMODEL coverage is used to define boundary conditions for the XMODEL finite element model that is under development through the USACEWES. Until the model has been tested and is available for general use it will not be documented as part of this reference manual.

CASC2D Coverage

The CASC2D coverage type is used when defining feature objects that are used to create grids (see the Create Grid command later in this section) for performing a rainfall/runoff analysis with the CASC2D finite difference model. Boundary conditions and model parameters can be assigned to the different feature objects and used in conjunction with the grid when saving CASC2D input files.

CASC2D Point/Node Types

CASC2D does not use Point attributes, but the dialog shown in Figure 3-12 is used to define following Node attributes:

Generic Generic nodes have no attributes, and are typically used at stream confluence.

Link Break CASC2D divides the grid cells defining channel segments up into links. Link breaks are automatically created at branching points, but may also be inserted at any node along the stream. This is typically done for long stream segments in order to keep the number of grid cells in a link to a reasonable number.

Weir Weirs can be defined at any node along a stream segment. When designating a node as a weir the CASC2D weir parameters should be defined.

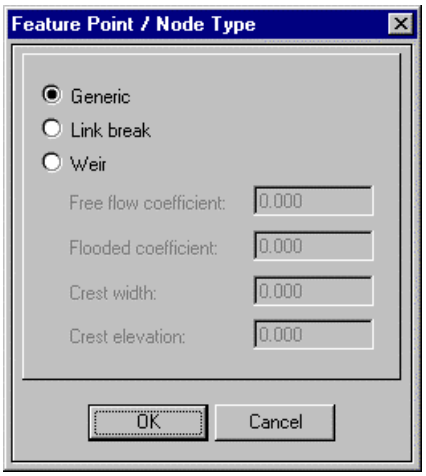


Figure 3-12 Feature Point/Node Type Dialog for CASC2D Coverages.

CASC2D Arc Types

The Arc Type dialog used for CASC2D coverages is shown in Figure 3-13. Arc attributes for CASCD include:

Generic Generic arcs have no attributes and are typically used when constructing polygons.

General Stream General stream arcs are identical to streams defined for drainage coverages and are used when going back and forth between coverage types.

Trapezoid Trapezoidal cross-sections are used when defining channel routing in CASC2D models. A Manning's N value, depth, bottom width and side slope must be defined for each arc of this type.

Break Point Break point cross-sections are cross-sections where area, conveyance, etc. for different locations of the cross-section.

Ridge Like general streams, ridge arcs are used for the purpose of compatibility with drainage coverages.

A Priessman slot is used to initialize channel flows prior to a CASC2D simulation. The slot is incised in the channel automatically from within the CASC2D model from the given width and depth parameters.

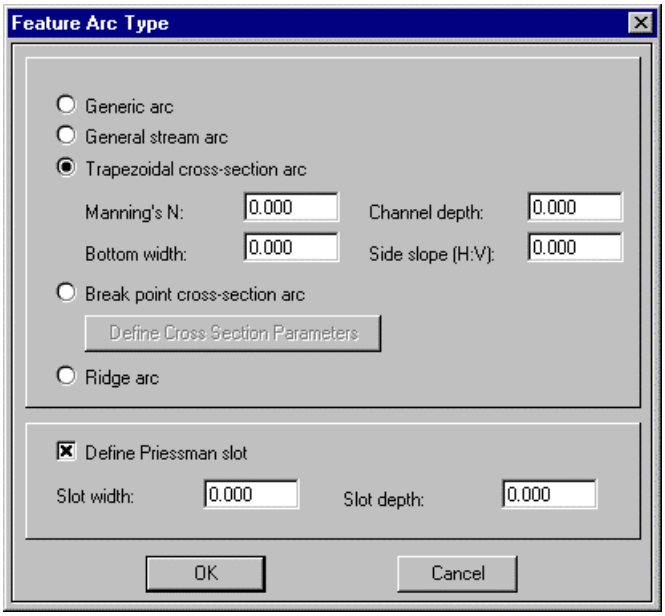


Figure 3-13 Feature Arc Type Dialog for CASC2D Coverages.

CASC2D Polygon Type

The Polygon Type dialog used for CASC2D coverages is shown in Figure 3-14. The polygon types are identical to the drainage coverage and include the following:

Generic Generic polygons have no attributes.

Boundary Boundary polygons are used to define the perimeter of the watershed and are used for creating grids from an already known watershed boundary.

Lake Lake polygons are used to define cells within a grid used to simulate lakes during a CASC2D simulation. Initial leakage discharge, spillway crest width, discharge coefficient, initial water elevation and crest elevation must be defined for each lake.

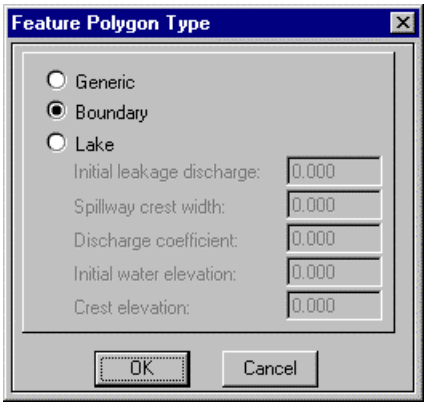


Figure 3-14 Feature Polygon Type Dialog for CASC2D Coverages.

3.2.7 Display Options

Display options control which feature objects are displayed. Each object is listed in the *Feature Object Display Options* dialog (Figure 3-15). The check box next to the object name can be toggled on or off to control whether or not the feature is to be displayed. In addition, the color button to the left of the check box can be used to set the color and other appropriate attributes such as radius, line thickness, line style, etc. Some of the display options are coverage specific and therefore the display options dialog is slightly different for each different coverage type. The active coverage type determines which of the display options dialogs is used.

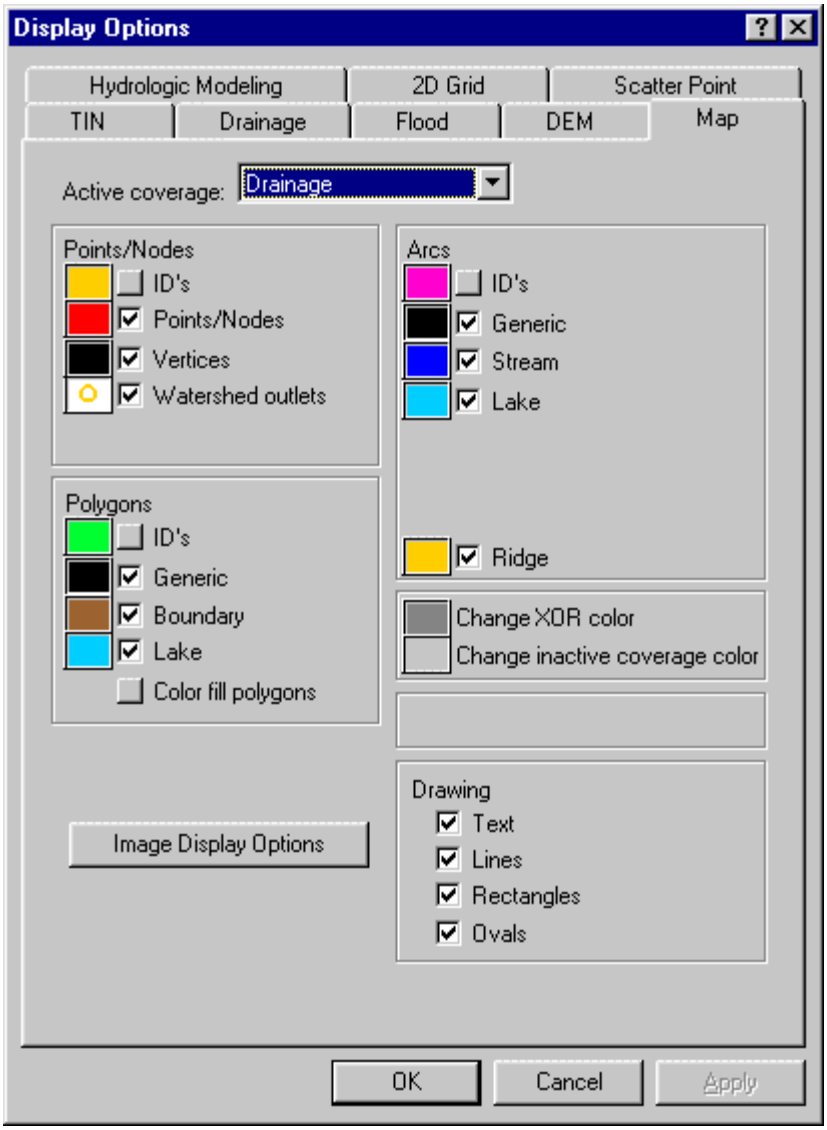


Figure 3-15 Feature Objects Display Options Dialog.

3.2.8 Menu Commands

There are several command available for creating, editing, and using feature objects to create stream networks, basins, and other supporting features. From these features TINs or grids may also be created which inherit important attributes defined on the feature objects. Many of the commands require that feature object(s) be selected prior to choosing the command. In such cases the tools described previously can be used for feature object selection.

Vertex<->Node

The *Vertex<->Node* command can be used to create a node from a vertex or a vertex from a node (providing the node is connected to only two arcs).

Vertices only define the geometry of the arc whereas nodes define the beginning and ending of an arc. Therefore when converting a vertex to a node the arc to which the vertex belongs is split into two separate arcs at the selected vertex. Likewise, when a node is converted to a vertex the two arcs attached to the node are merged into a single arc (if more than two arcs are attached to a node it cannot be converted to a vertex).

Cleaning Feature Objects

The *Clean* command is used to clean up feature object data. Specifically, it prompts for a snapping tolerance and minimum dangling arc length, and then uses these parameters to do the following:

1. A check is made to see if any nodes are within tolerance of other nodes. If so, the nodes are snapped together.

2. A check is made to see if any arcs intersect. If so a node is created at the intersection and the arcs are split.

3. A check is made for dangling arcs (arcs with one end not connected to another arc) with a minimum length. If any are found they are deleted.

All objects of the active coverage will be cleaned.

Alternatively, a check is made to see if any arcs, vertices, nodes, or points are selected in the active coverage. If any intersecting arcs are selected, you have the option of intersecting only the selected arcs. If any points, nodes, or vertices are selected, you snap the selected points, nodes, and vertices to a snapping point that is selected after the *Clean Options* dialog is closed.

Translating Points/Vertices

The *Translate* command can be used to translate all selected feature points/vertices a given x and y distance. Sometimes points imported from DLG files or digitized stream files do not overlay the background elevation map as they should (this can usually be discerned by displaying contours of the DEM or TIN serving as the background elevation map). In such cases the points/vertices may be adjusted by first selecting points or vertices to be panned using the appropriate tool and then specifying an x and y translation value.

Locate Streams/Ridges

The *Locate Streams/Ridges* command can be used to automatically move the xy position of the nodes and vertices of ridge and stream arcs so that it overlays the lowest elevation "near" the point on the background elevation map (either a DEM or a TIN). The search for an elevation "near" the point moves in a direction perpendicular to the arc.

The distance criteria (set in the dialog shown in Figure 3-16) determines how far the search continues looking for a lower elevation. For DEM's you may specify either the number of grid cells, the average length of the two adjacent arc edges, or a specified distance. For TINs the number of closest DEM cells is dimmed. When a node or vertex is moved by the maximum amount it is highlighted and a warning message is given.

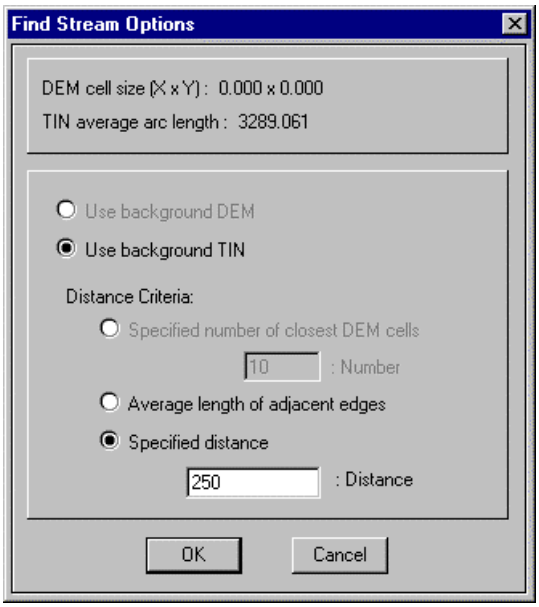


Figure 3-16 Find Stream/Ridge Options Dialog.

Reordering Streams

The *Reorder Streams* is used to make sure that the direction of all stream arcs are consistently defined from a selected outlet node. When creating stream arcs in *WMS* a check is made to ensure that points are created from downstream to upstream, so that the first node in the arc is the downstream node. This makes it impossible to create streams in *WMS* that do not follow this definition. However, when arcs are imported it is possible to create streams where the downstream node is second rather than first for some arcs but not others. All arcs can be consistently ordered by selecting an outlet node (a node attached to only one arc) and then choosing the *Reorder Streams*.

Redistributing Vertices

Vertices along arcs can be redistributed at either a higher or lower density using the *Redistribute* command in the *Feature Objects* menu. The vertex density along arcs determines the density of TIN vertices when issuing the *Create TIN* command from the *Feature Objects* menu. Vertices are redistributed along all selected arcs using a uniform or cubic spline method.

Arcs can be selected one at a time using the *Select Arc* tool, the *Select All* command in the *Edit* menu (the *Select Arc* tool must be active), the *Select Branch* tool, or the *Select Network* tool. Once the arcs are selected the method of redistribution can be chosen from the *Redistribute* dialog shown in Figure 3-17.

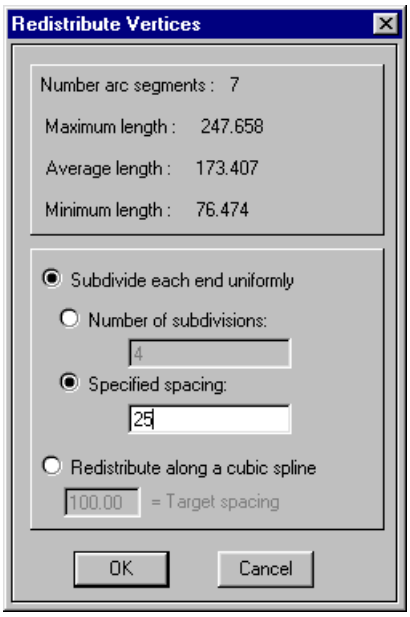


Figure 3-17 Redistribute Vertices Dialog.

Uniform Subdivision

If the *Subdivide each end uniformly* options is specified then either a number of intervals, or a specified spacing can be given to determine how points are redistributed along the selected arcs. If the specified spacing is greater than the length between adjacent arc vertices, the vertices are moved to reflect this larger spacing.

Spline Redistribution

If the *Redistribute along a cubic spline* option is specified vertices between arcs are redistributed by creating a series of splines from the vertices of selected arcs and then redistributing a new set of vertices at the specified distance. If the flag for preserving points with an angle greater than the specified angle is checked then vertices at such locations will remain after the redistribution. This method can be used to create vertices at a smaller density (specify a larger target spacing) as well as a higher density.

Figure 3-18-Figure 3-21 illustrates the difference between these different methods. Figure 3-18

Figure 3-18shows a set of arcs after their initial creation. Figure 3-19 shows redistribution with a constant interval of 5 (5 vertices between the beginning and ending of each arc. Figure 3-20 shows uniform redistribution with a constant spacing of 100 (meters in this instance). Figure 3-21 shows redistribution along cubic splines. It can be seen that the uniform redistribution at a constant spacing and redistribution along cubic splines are very similar, whereas the uniform redistribution using a number of intervals results in a much different result. In all cases redistribution of vertices along the arcs was done by first selecting all arcs and then redistributing along all arcs with a single method. The uniform redistribution with a constant interval would likely work well provided it is done on an arc by arc basis.

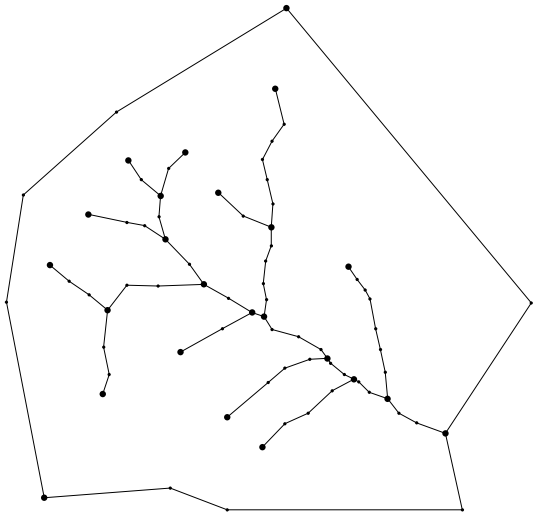


Figure 3-18 Arcs Before Redistribution.

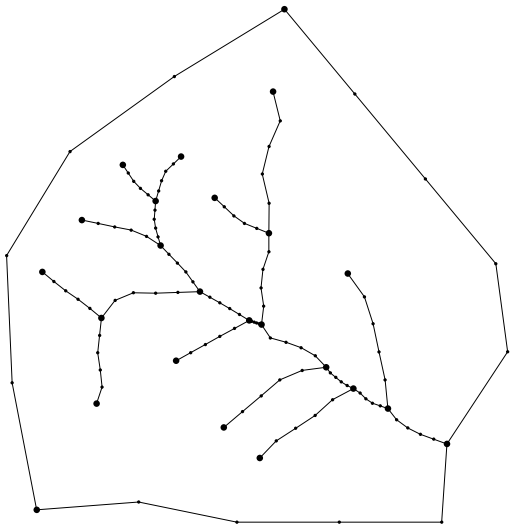


Figure 3-19 Uniform Redistribution with Constant Interval of 5.

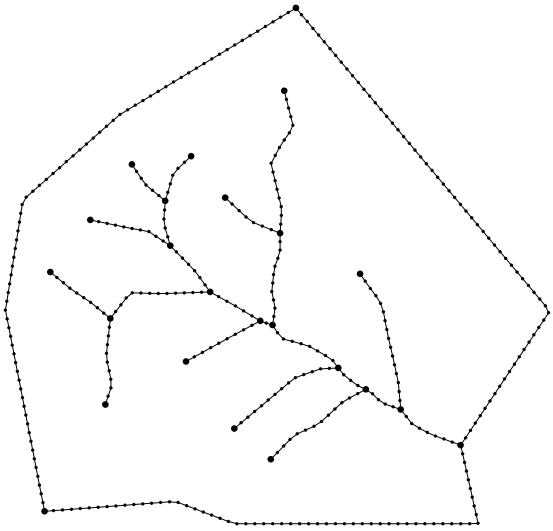


Figure 3-20 Uniform Redistribution with Constant Spacing of 100.

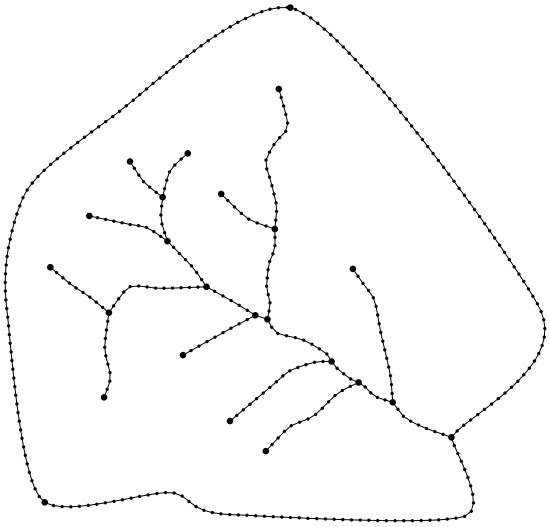


Figure 3-21 Spline Redistribution with Spacing of 100.

Deleting Isolated Points

Many times the density of digitized points, or points imported from a DLG file is much higher than is needed to create a boundary polygon or set of stream arcs. The *Delete Isolated Points* command can be used to eliminate all points not attached to arcs.

Reversing the Direction of Arcs

The *Reverse Directions* command can be used to reverse the direction of selected arcs. This is used for stream type arcs where the direction of connectivity is important. Stream arcs must always be defined by connecting point from downstream to upstream. If an error was made when creating the points this command can be used to correct it. An arrow is drawn on all arcs from upstream to downstream and can be used to verify that directions are correct.

Adding Arcs to TINs

A set of arcs can be inserted as breaklines into an existing TIN using the *Arcs->Breaklines* command. Inserting a breakline forces triangle edges to conform to the arc. New TIN vertices are created for all nodes and vertices of the arcs and elevations for these new vertices are assigned from the z value of the nodes/vertices.

Building Polygons

Just defining a series of arcs that form a closed loop, or polygon, does not create a polygon. Polygons are created from arcs only after the *Build Polygons* command is used. Feature polygons can be created in one of two ways:

1. If there are no selected arcs when the *Build Polygons* command is chosen, polygons are created for all arcs of the active coverage that form closed loops, or polygons. The one exception being that if the active coverage is a “drainage” type coverage, stream arcs are not used to create polygons. If you want a stream arc to also form a basin boundary then you must build polygons according to method two below.

2. If there are selected arcs when then *Build Polygons* command is chosen then polygons are only created for closed loops or polygons formed by the set of selected arcs.

By either method, the new polygon inherits the current default polygon type, unless in the drainage cover and then the new polygon will be defaulted to a lake polygon if all it's arcs are lake arcs and a drainage polygon otherwise.

Creating TINs

A TIN can be created from a set of feature objects in a drainage coverage using the *Create TIN* command. The density of vertices in the TIN will be proportional to the vertex spacing along arcs. The *Redistribute* command in the *Feature Objects* menu can be used to adjust vertex spacing and locally refine the TIN in important areas. Either an existing TIN or a DEM can be used as a background elevation map when interpolating z values for the

vertices of the TIN. If appropriate z values have been assigned to the feature arcs then the z values from the arcs will override z values interpolated from the background elevation map for TIN vertices created from feature arcs vertices. Figure 3-22 shows a TIN which results from creating a TIN using the feature objects shown in Figure 3-21

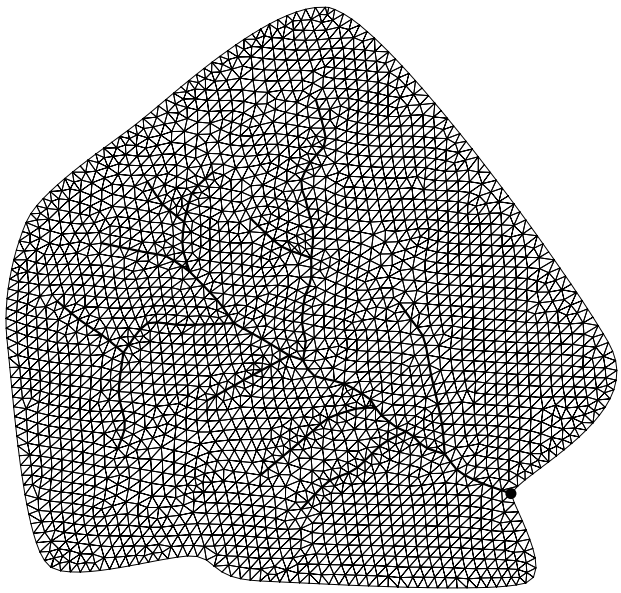


Figure 3-22 TIN Created from Feature Objects.

For TINs requiring a lot of memory (high resolution of vertices or covering a large spatial extent), it may be advantageous to build the TIN in the absence of a background DEM. Interpolation of elevations to the TIN from the DEM afterwards can be done in a "block by block" fashion using the *Interpolate to TIN* command found in the *DEMs* menu. In other words you can read in portions of the DEM and interpolate to TIN multiple times. Elevations for TIN vertices that are not within the extents of the current DEM are not interpolated. No such option exists if a TIN is used as the background elevation source.

If you already have TIN data why would you ever use a TIN as a background elevation set? The primary purpose of creating TINs from feature objects is to insure that stream channels and other important hydrologic features are adequately represented in the TIN as triangle edges. If you simply triangulate a set of xyz scatter points, or import a TIN from another data source, it is not likely that this condition will exist. Creating a new TIN from feature objects will insure that the TIN is optimal for performing drainage analysis because the new TIN will be built "around" the feature objects.

Creating Grids

A grid can be created from a feature polygon using the *Create Grid* command. Active and inactive cells are determined from the boundary polygon. A rectangular grid is created that encompasses the bounds of the boundary polygon and cells outside the polygon are assigned an inactive status. Either an existing TIN or a DEM can be used as a background elevation map when interpolating z values for the elevation data set of the grid. An example of a grid created from the feature objects of Figure 3-21 is shown in Figure 3-23.

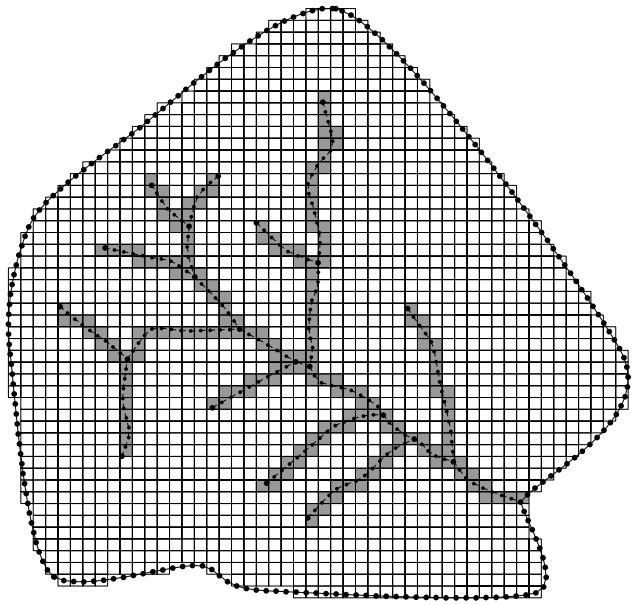


Figure 3-23 Grid Created from Feature Objects.

Importing Feature Points

Feature points can be imported from two different sources: Digital Line Graph (DLG) files and digitized points. Imported points which are not connected together to form arcs become points. They can later be connected together using the *Create Arc* tool to form arcs.

Deleting Feature Objects

Feature objects can be deleted by selecting the feature object(s) to be deleted and hitting the *DELETE* or *BACKSPACE*, or by selecting the *Delete* command from the *Edit* menu. If the *Confirm Deletions* option in the *Edit* menu is active, you will be prompted to confirm each deletion. This is helpful in

preventing accidental deletions. The confirm deletions flag can be toggled by selecting the *Confirm Deletions* item.

Changing Feature Points/Nodes/Vertices Positions

Two methods of editing feature points/nodes/vertices positions and z values are available. To manipulate positions and z values, the appropriate select tool (points/nodes or vertices) tool must be selected.

- A feature point/node/vertex can be moved to a new position by clicking on it and holding down the mouse button while dragging it to the desired position.

- The feature point/node/vertex position can also be modified by selecting the point and changing the xy values that appear in the x and y edit boxes.

Updating Basin Data

When the drainage coverage feature objects are used to create a hydrologic model the area of the polygons can be determined and used in any of the supported hydrologic models. If the points/vertices used to create the feature arcs also have z values associated with them then *WMS* will make a rought estimate of the longest flow path and slope along such a path. Finally, if you have derived a watershed from feature objects, but have a background DEM you can compute most of the basin data parameters using the following steps:

1. *Import/Read* the DEM.

2. Compute the flow directions using *TOPAZ* (see section 5.4).

3. Use the *Polygon Basin IDs -> DEM* command found in the *Drainage* menu in the *DEM* module to assign DEM cells a basin id from the feature object polygons.

4. Choose the *Update Basin Data* command from the *Feature Objects* menu.

Of course the results will only be approximate since the actual basin boundaries will not have been derived from the computed flow direction data, but it will provide a reasonable estimate. Ideally you will want to derive the basin boundaries from the DEM and flow direction data and then use the *Compute Basin Data* command in the *DEM* module (see section 5.5).

Creating Time Computation Arcs with TINs and DEMs

The *Node->Flow Arcs* command allows you to automatically develop time computation arcs by tracing the flow path across a TIN or DEM and saving an arc representing the path so that an equation can be assigned to it (see pages 3-

18, and 15-5 for more information on using time computation arcs to compute time of concentration and lag time). You can investigate where the arc(s) will be created using the flow path tool and selecting points on the TIN or DEM since the same functions for flow are used to create the arcs. The process requires the following three steps:

1. Create a feature points at locations where the flow path segments should begin.

2. Make sure any feature points you wish to use to create time computation arcs are selected.

3. Choose the *Node->Flow Arcs* command.

You will be asked if you want to create one continuous arc or multiple arcs. If you respond with one continuous arc it will create an arc representing the flow path from the selected feature point to the next downstream outlet. If you choose to create multiple arcs it will break the arc into separate arcs if the flow path encounters a stream.

Converting Stream Arcs to Time Computation Arcs

When using the *Node->Flow Arcs* command as described above, only the portions of the stream that are part of the flow path from the selected point to the outlet get converted to time computation arcs. If you wish to compute the lag time between consecutive outlet points then you will need to convert the remaining stream portions to time computation arcs. This is the purpose of the *Streams->Flow Arcs* command. Not all streams are converted when using this command, only those stream segments that connect outlets. This command works for either TIN or feature object stream segments.

3.2.9 CASC2D

CASC2D is a two-dimensional finite difference rainfall/runoff model. A finite difference grid is used to establish the computational domain and parameters for surface runoff. The *CASC2D* model is fully coupled with hydraulic stream flow/routing models. Parameters for stream channels are defined using arcs and then mapped to the appropriate underlying grid cells.

In order to define *CASC2D* channel parameters using arcs, the current coverage type must be set to *CASC2D*.

Smoothing Stream Cells

Because elevation data used to define the surface runoff component of *CASC2D* does not contain the detailed resolution required to capture the actual stream bed elevation of the channels, the bed elevation profile can be highly irregular. These irregularities, or abrupt changes in elevation can cause

instabilities in the channel routing computations, and therefore must be smoothed out.

The *Smooth Stream Cells* command is used to adjust the elevation of the stream bed for *CASC2D*. Smoothing is done by first selecting a continuous set of arcs which represent the stream and then choosing the *Smooth Stream Cells* command from the *CASC2D* menu. The dialog shown in Figure 3-24 is then used to smooth the channel bed elevation.

Initially the bed elevation is assigned the same value as the grid. Whenever a profile is shown, the bed elevation profile is displayed in blue while the grid elevation of the cell is shown in red. Care should be taken to see that no bed elevation is higher than the grid cell elevation.

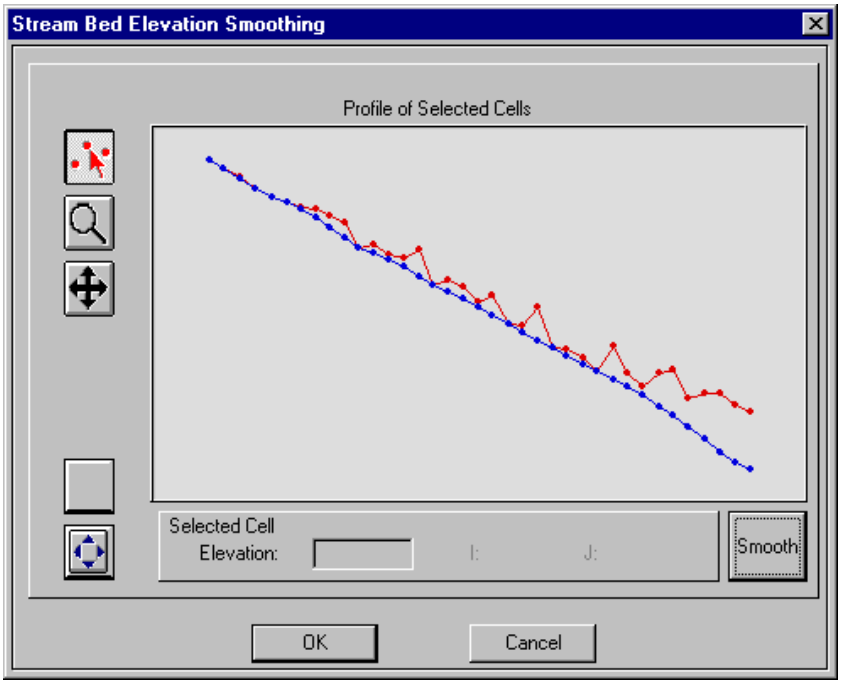


Figure 3-24 Smooth Stream Cells Dialog.

Renumber Links And Nodes

In order to properly execute the channel routing routines of *CASC2D*, the stream channel must have the proper order and connectivity. This ordering or numbering can be done automatically using the *Renumber Links and Nodes* command from the *CASC2D* menu.

Links define whole channel segments and must be numbered such that any segment has no "upstream" segments with a link number that is greater than itself. In other words all channel segments must "flow" into downstream segments with a higher link number.

3.3 Drawing Objects

Drawing objects are used to enhance or provide annotation to a model. For example, text may be used in conjunction with an arrow identifying a key aspect of the model, or rectangles and ellipses may be added to highlight some feature of the model. Drawing objects are saved as part of the map file and are restored when that file is read. The types of drawing objects available are text, lines, rectangles, and ovals, and are created using the tools in the dynamic palette as described in this section.

3.3.1 Drawing Object Tools

The following drawing object tools are in the dynamic portion of the *Tool Palette* when the *Map* module is activated. Only one tool is active at any given time.

Create Text Tool

The *Create Text* tool can be used to annotate objects in the *Graphics* window prior to printing and inclusion in report documents. The location clicked on defines the beginning point on the screen where the defined text string will be placed. After clicking on a location the *Text Attributes* dialog will appear allowing you to define the text string, font, and color. A rectangle the color of the background can be used to erase the area behind the text.

Create Rectangle

The *Create Rectangle* tool can be used to create wire frame or filled rectangles anywhere on the display. Rectangles can be used to represent buildings, frame text strings, etc., in order to enhance a printed file to be used in reports. Rectangles are created with this tool by dragging a rectangle with the mouse at the location on the screen where you wish to place the rectangle.

Create Oval

The *Create Oval* tool can be used to create wire frame or filled ellipses anywhere on the display. Ovals can be used to represent buildings, frame text, etc., in order to enhance a printed file to be used in reports. . Ovals are created with this tool by dragging a rectangle with the mouse at the location on the screen where you wish to place the oval. The rectangle width and height determine the major and minor axes of the oval.

Create Line

The *Create Line* tool can be used to create a line anywhere on the model. Lines can be used to identify key locations on a model, or in conjunction with

labeling text to enhance printed images. A line is created by clicking out a polyline on the screen with the mouse and double-clicking to end. The attributes of lines include color, style, width, and an arrowhead at the beginning or ending of the line.

Select Drawing Objects

The *Select Drawing Objects* tool can be used to select previously created text, rectangles, ovals, and lines. Once selected, a drawing object can be moved to another location by dragging with the cursor while the select drawing object tool is active. Attributes of an object may also be changed by first selecting the object and then choosing the *Attributes* command from the *Drawing Objects* menu. The drawing objects attributes are discussed in the next section.

3.3.2 Display Attributes

When a new object is created it inherits the current attributes for that object type. However, any object may be selected, and it's attributes changed by selecting the *Attributes* command from the *Drawing Objects* menu, or by double-clicking on the object with the select drawing object tool active.

Text Attributes

The text attributes include a font and a color. These can be changed using the text attributes dialog shown below.

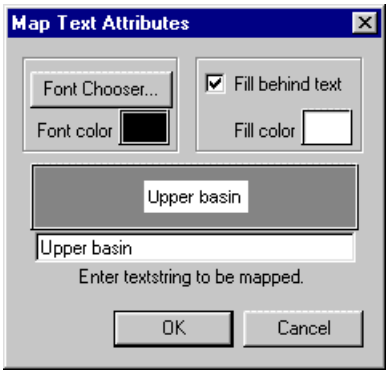


Figure 3-25. Text Attributes Dialog.

Rectangle and Oval Attributes

Rectangle and oval attributes include whether or not it is filled, a fill color, a fill style, and a line (border) color, style, and width. These attributes can be edited with the *Rectangle/Oval Attributes* dialog box shown below.

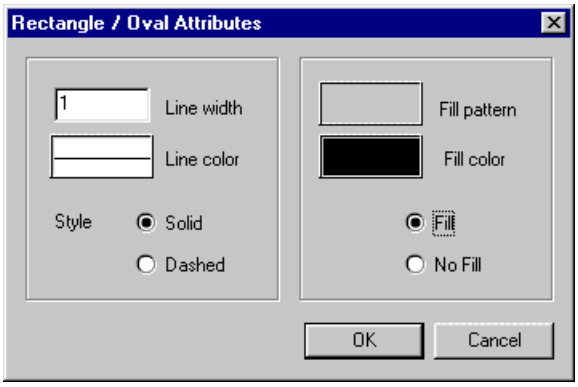


Figure 3-26. Rectangle and Oval Attributes Dialog.

Line Attributes

Line attributes include a color, style, and width as well as whether or not the beginning, ending or both of the line contain arrowheads. These attributes are edited with the *Line Attributes* dialog shown below.

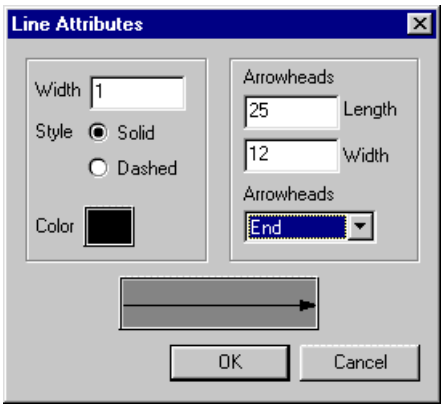


Figure 3-27. Line Attributes Dialog.

3.3.3 Drawing Depth

The drawing depth of drawing objects can make them seem closer, or farther away in relation to other visible objects of your model. By default, drawing objects are drawn at the average depth (z value) of visible objects in your model. However, an explicit z value can be set using the *Drawing Depth* command. In plan-view the drawing depth will have no effect on the overall appearance of the objects.

3.3.4 Drawing Order

The order in which drawing objects are displayed becomes important whenever a rectangle or oval is displayed in color fill mode. For example if a

rectangle is used to erase an area behind a text string it must be drawn prior to the text string or it will result in erasing the text string as well. The order of drawing objects can be controlled using the *Move to Front*, *Move to Back*, *Shuffle Up*, and *Shuffle Down* commands described below.

Move to Front

The *Move to Front* command causes the selected drawing object to be drawn last. In other words it will appear on top or in front of all other drawing objects.

Move to Back

The *Move to Back* command causes the selected drawing object to be drawn first. In other words it will appear at the bottom or in back of all other drawing objects.

Shuffle Up

The *Shuffle Up* command causes the selected drawing object to be displayed one object later than it is currently be displayed. This causes it to appear in front of the object that is currently being displayed just ahead of it.

Shuffle Down

The *Shuffle Down* command causes the selected drawing object to be displayed one object sooner than it is currently be displayed. This causes it to appear in back of the object that is currently being displayed just behind it.

3.4 Images

An image is a digital picture that has been scanned from a photo or captured from the screen of another application. A common format for saving images is the TIFF (Tags Image File Format) format. *WMS* supports reading and writing of TIFF images for the purposes of draping on a TIN or grid, or displaying in the background when performing on-screen digitization of feature objects or TIN vertices.

Images must be registered to the TIN, grid, or background by defining real world coordinates to positions on the image. This establishes a mapping between real world coordinates and the pixels of the image. The most important part of using an image in *WMS* is to properly register it. The image itself consists of a two-dimensional array of screen pixel colors. However, the image does not contain information about the real world coordinates which correspond to the extents of the image. For this reason the xy pixel locations of the image must be mapped to user-specified real world coordinates. For example if an image of a 7.5 minute quad sheet is scanned and you wish to use

it in *WMS* as a background map for digitizing, then the xy coordinates of the quad corners must be registered to the xy pixel limits of the TIFF image. If this is not done correctly then any points created by on-screen digitizing, using the background image as a guide, will have incorrect coordinates.

Once an image has been imported and registered, the image file name and registration points can be saved as part of the super file and read back in with the model. An image file with the name of the TIFF file and the registration points can also be exported using the *Export* command found in the *Image* menu. These files can then be read in together by opening the image file using the *Open* command in the *File* menu.

3.4.1 Display Options

The Image *Display Options* (Figure 3-28) control whether the currently registered image is displayed, and whether it is registered as a background image, or draped over a TIN or a grid.

There are two different ways an image may be mapped and used in *WMS*:

- **Draw on XY plane behind all objects** - In this case the image is registered to specified xy coordinates and drawn first so that it appears behind all other objects.

- **Texture mapped to a TIN or GRID** - In this case the image is "draped" over the active TIN or GRID. The image must be registered to an xy location within the domain of the TIN or GRID.

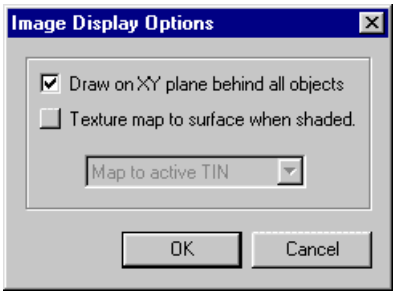


Figure 3-28 Image Display Options Dialog.

3.4.2 Importing Images

The *Import Image* command is used to read a new image. It is only possible to have one image at a time in *WMS*, so reading a new image will always cause an existing image to be deleted. When this command is issued, the *File Browser* is used to select the TIFF image file, and the image is drawn in the *Register Image* dialog as shown in Figure 3-29 and explained in section 3.4.3. Some of the new TIFF images contain registration information within the

TIFF file itself (they are often called georeferenced TIFF files). If you import a georeferenced TIFF file you will not need to go to the registration window and set up the registration points manually.

3.4.3 Registering An Image

As described above, an image must be properly registered in order to be used accurately in combination with other data objects, or as a background for on-screen digitization. Registering an image is accomplished by specifying the real world and corresponding image (pixel) coordinates of three different points. The dialog in Figure 3-29 is used to register an image after it has been imported, or it may be used to change or correct the registration points of the current image. In this dialog the image is displayed in a graphics window with three X's representing the registration points. The real world coordinates (x,y) and image coordinates (u,v) of the three registration points are listed in edit fields below. Typically you will know the actual coordinates for a few features on the map. The easiest way to register the points is to drag the points (using the tools described below) to known points on the image (this modifies the u,v coordinates) and then enter the xy or real coordinates of that point. Since the image can take a few seconds to update in the register window, the Escape key can be used to abort the update after enough of the image is visible to see where you want to move a registration point to.

The Lat/Lon calculator buttons can be used to convert a latitude-longitude pair into equivalent UTM coordinates.

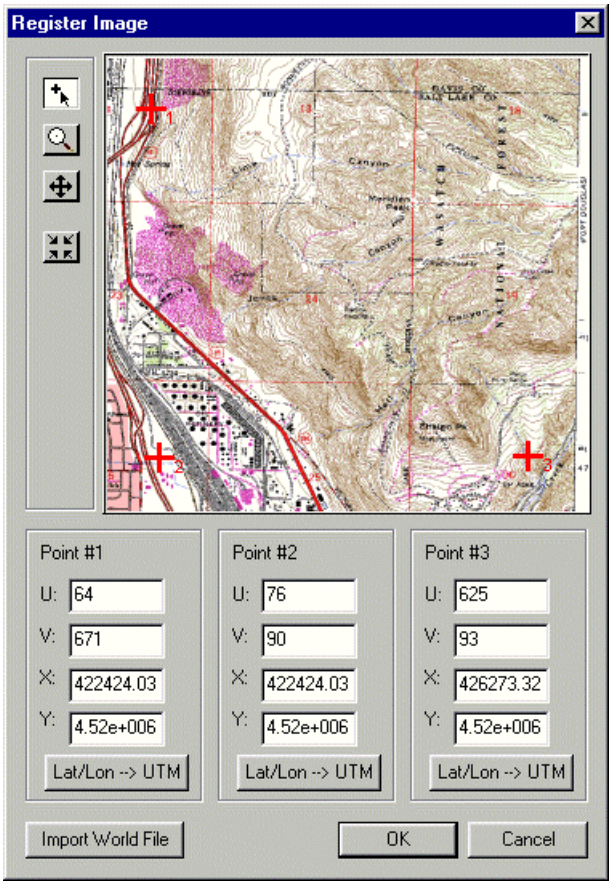


Figure 3-29 Register Image Dialog.

The *Import World File* button can be used to register the image from an *ARC/INFO®* world file. The world file contains the same information as the *WMS* image registration file and can be used by *WMS*. You will also note in the export option that you can also save a world file from *WMS* to register an image in *ARC/INFO®* or *ArcView®*.

Register Point Tool

The *Select Register Point* tool is used to select and drag register points to a location on the map for which real coordinates are known so that they can be entered in the corresponding xy edit fields.

Zoom Tool

The *Zoom Tool* is used to zoom in a portion of the image for which coordinates are known and a registration point needs to be placed.

Pan Tool

The *Pan Tool* is used to pan the image to a location for which coordinates are known and a registration point needs to be placed.

Frame Macro

The *Frame Macro* is used to automatically center the image within the drawing window of the dialog after panning and zooming in on a specific location.

3.4.4 Resampling Images

When a TIFF file is imported the colors for each pixel are read into memory. The image itself will often have a higher resolution than the computer screen you wish to display it on. For example and image might be 1600x1600 pixels where as your computer monitor may only be 800x800. This means that the entire image can not be viewed at full resolution. Initially, *WMS* resamples the image so that it can fit the entire image on the computer screen at the best possible resolution. For example, using the image and screen size above only one in four pixels (every other one in each direction) from the image can be displayed since the image has 4 times as many pixels as the computer screen.

If you zoom in on the image you will notice that each image pixel appears larger than one screen pixel. This occurs because when the image is resampled there is one pixel from the image for every screen pixel, but after zooming there are fewer pixels from the image (in the zoom region) than there are screen pixels. The image can be resampled to fit the screen again by selecting the *Resample* command from the *Image* menu.

Immediately after resampling the image you cannot view any part of it that is outside of the screen. However, if you zoom without resampling, you can pan the image to view parts of it which are now outside of the screen.

3.4.5 Fit Entire Image

The *Fit Entire Image* command fits the boundary of the image within the screen and shows the currently resampled region of the image. This is useful when you wish to orient where you are viewing relative to the entire or image, or if you wish to zoom in on another portion of the image without having to resample the entire image first.

3.4.6 Deleting Images

Deleting the image removes it from memory. Since only one image at a time can be displayed, the current image must be deleted whenever a new image is either imported or created using the screen capture command.

3.4.7 Exporting Images

The *Export Image* command is used to save a TIFF image and its accompanying registration points. When this command is issued the currently contents of the TIN window are written out in a TIFF file format, and the TIFF file name and registration points are saved in an image file that can be reopened later using the *Open* command from the *File* menu. This command can be used to save the contents of a complex display (one that may be requiring several seconds to update) to an image. It can then be used as a "backdrop" or base map and the image will update much faster than the original display.

Besides saving the TIFF image and registration image file, an *ARC/INFO*® world file may be saved. World files are used by *ArcView*® and *ARC/INFO*® for displaying registered images as well as several other programs with image display capabilities.

3.4.8 Capturing The Screen

Displays created by *WMS* can be captured to a TIFF image file using the *Screen Capture* command. These images can then be used in *WMS* as backgrounds, or can be used as images in other applications or report documents.

When the *Screen Capture* command is issued the image currently in the *Graphics* window is converted internally to a TIFF image. Since the real coordinates of the screen corners are already known the image is automatically registered as it is captured. These images and registration points can be exported and read back into *WMS* at a later time.

3.5 DXF Files

The Drawing Exchange Format (DXF) is a file format commonly used by CAD programs. DXF files can be imported in *WMS* to provide a background map of rivers and other terrain features, or to enhance the final output image of a modeling project.

3.5.1 Display Options

The Display options dialog is shown in Figure 3-30 and includes options to make all objects visible, hide all objects, and delete all objects. In addition, individual objects can be selected or unselected by clicking on the object in the text window display. An asterisk (*) is placed to the left of all selected objects. Colors and styles of objects in the selected layer may also be controlled by selecting the appropriate color box.

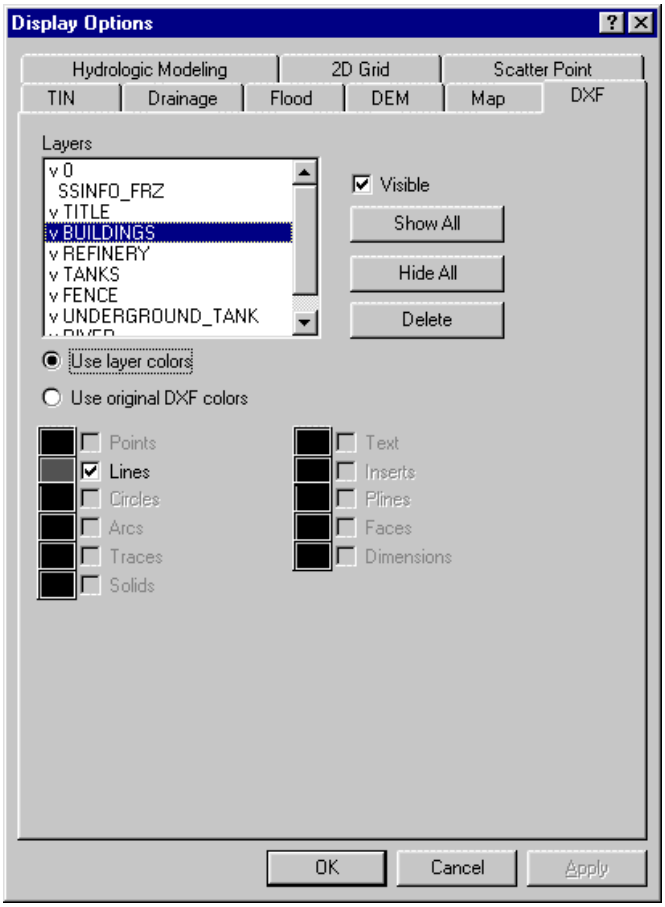


Figure 3-30 DXF Display Options Dialog.

3.5.2 Importing DXF Files

The Import DXF files command is used to read a DXF file. Once a file is read all objects are visible. DXF objects can be unselected (hidden) using the *DXF Display Options* dialog.

3.5.3 Deleting DXF Objects

The *Delete* command is used to delete all DXF objects. Deleting can also be done from within the *Display Options* dialog.

3.5.4 Converting DXF Objects to Feature Objects

Many times you will want to use your DXF data to create streams, watersheds, or other feature objects. To do this you must first convert the DXF data to feature objects. This is done by choosing the DXF -> Feature Objects command and then specifying which objects in the layer you want converted from the dialog shown in Figure 3-31. When converting data you may either

add it to the currently active coverage, or have a new coverage created. Once the data have been converted it is a good idea to delete the DXF data.

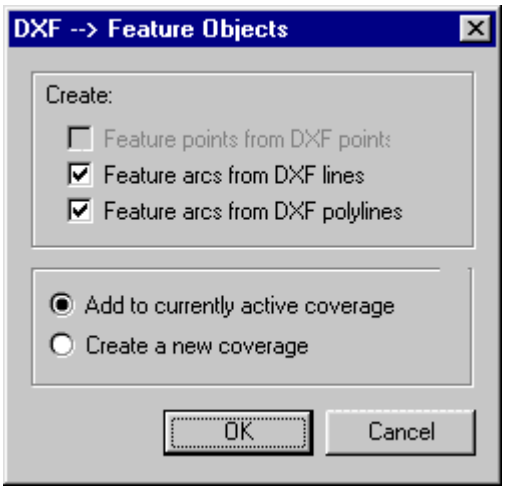


Figure 3-31 DXF->Feature Objects Dialog.

3.5.5 Converting DXF Objects to TINs

Sometimes digital elevation data are stored in DXF files in the form of 3D points and 3D faces. These DXF objects can be converted to TINs using the DXF->TINs command in the DXF menu. After converting the data it is a good idea to delete the DXF data.

CHAPTER 4

DEMs

4.1 Introduction

Because of the widespread availability of elevation data in gridded format, several tools have been implemented in *WMS* so that gridded elevation data can be used for automated watershed delineation and hydrologic analysis. The United States Geological Survey (USGS) provides 1:250,000 scale digital elevation models (DEMs) for the entire U.S. and has 1:24,000 scale DEMs for most areas. This data can be downloaded by using Netscape, Internet Explorer or any other browser from the USGS web site at the following URL: http://edcwww.cr.usgs.gov/doc/edchome/ndcdb/ndcdb.html.

In addition to USGS DEMs, grids processed from *ARC/INFO*® or *GRASS* can also be imported. DEM's can be contoured and used as a guide for the placement of boundary, stream, and ridge lines.

This chapter focuses on the general uses of DEMs including display and editing prior to use for basin delineation. The next chapter focuses on the use of DEMs for automated basin delineation and hydrologic parameter calculation.

4.2 Tool Palette

The following tools are in the dynamic portion of the *Tool Palette*. These tools are available when the *DEM* module is activated. Only one tool is active at any given time. The action that takes place when you click in the *Graphics*

Window with the cursor depends on the current tool. There are several other tools intended for use when doing drainage analysis with DEMs that are described in the next chapter.

4.2.1 Select DEM Points

The *Select DEM* point tool is used to select a region of DEM points to make active or inactive. Coordinates of DEM points may not be edited. When selecting a group of DEM points a rectangle or polygon around the points is displayed rather than trying to identify individual DEM points. To deselect a group of DEM points that have already been selected, click anywhere in the graphics window. Multiple groups of DEM points may be selected by holding down the SHIFT key while dragging a rectangle around the second group.

4.2.2 Contour Labels

The *Contour Label* tool manually places numerical contour elevation labels at points clicked on with the mouse. These labels remain on the screen until the contouring options are changed, until they are deleted using the *Contour Label Options* dialog, or until the *Graphics Window* is refreshed. Contour labels can be deleted with this tool by holding down the *SHIFT* key while clicking on the labels. This tool can only be used when the DEM is in plan view.

4.3 DEM Display Options

Display options control which features of the DEM are displayed. Each display feature associated with DEMs is listed in the *Display Options* dialog (Figure 4-1) under the DEM tab. DEM display options are accessed by selecting the *Display Options* command in the *Display* menu. The check box next to the feature name can be toggled on or off to control whether or not the feature is to be displayed. In addition, the color button to the left of the check box can be used to set the color and other appropriate attributes such as radius, line thickness, line style etc. General purpose display options are documented in this chapter and the rest of the DEM display options are documented in the next chapter on drainage delineation with DEMs.

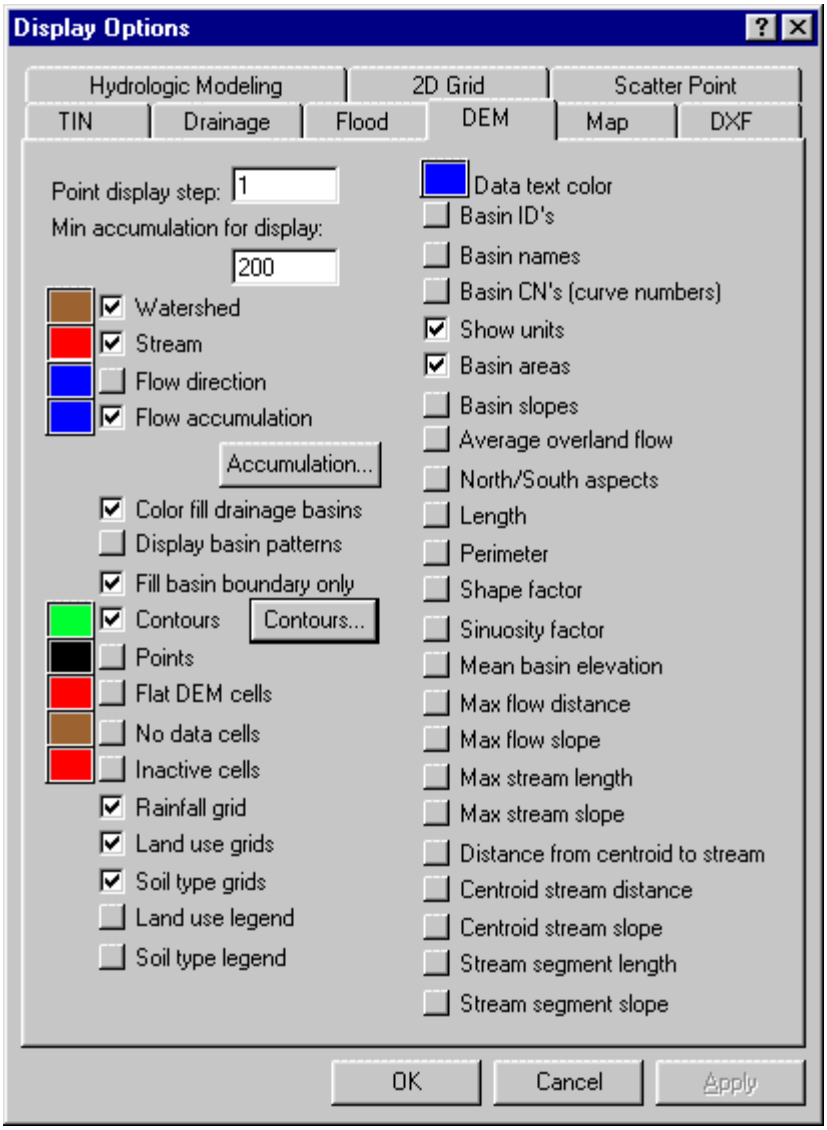


Figure 4-1 DEM Display Options Dialog.

4.3.1 Points

If the *Points* item in the *DEM Display Options* dialog is set, the DEM points will be displayed each time the *Graphics Window* is refreshed. The *Display Step* can be used to control the density of points that are displayed. The color of DEM points is used when performing a hill shade using the *Shade* command.

4.3.2 Contours

If the *Contours* item is set, the DEM will be contoured according to the options set in the *Contour Options* dialog accessed from the *Display* menu.

Contouring a DEM can be valuable in determining where streams and other feature arcs should be placed.

4.3.3 Flat DEM Cells

If the *Flat DEM Cells* item is set, all flat DEM Cells will be displayed each time the *Graphics Window* is refreshed. A DEM cell is considered flat if four adjacent DEM points all have the same elevation. Adjacent DEM points are determined using the *Display Step* as defined below.

4.3.4 NODATA Cells

A DEM point or cell is a NODATA cell if an elevation value for that point does not exist. A DEM must be rectangular since elevations are accessed as a elements of a two-d array. If the domain of valid DEM points is not perfectly rectangular then a rectangular bounding box is placed around the valid points and points inside the bounding box but not within the valid domain take on a NODATA status.

4.3.5 Inactive Cells

Inactive cells have valid elevations but are temporarily disabled. Their display can be seen by toggling this option on.

4.3.6 Land Use

A separate land use ID grid, with a different resolution than the elevation DEM, may be used in *WMS*. A land use id may be assigned to each DEM point. This id can then later be related to a table referencing curve numbers for different soil types and is used in the calculation of a composite curve number. A legend for land use may also be toggled on.

4.3.7 Soil Type

A separate soil ID grid, with a different resolution than the elevation DEM, may be used in *WMS*. A soil type id may be assigned to each DEM point. This id can then later be related to a table referencing soil type information (typically hydrologic soil group classification). Soil types are used in the calculation of a composite curve number. A legend for soil type may also be toggled on.

4.3.8 Display Step

If a large number of grid points are imported, the display can become too cluttered or the display time too lengthy. The *Display Step* can be adjusted to

speed up and/or improve the visibility of what is displayed. However, if the display step is larger than one, contours are interpolated from fewer points and will appear less accurate Nevertheless, the display step is ignored when interpolating z-values to grids or TINs.

4.4 Importing DEMs

DEM data from USGS (standard format and SDTS), *ARC/INFO®*, *DTED*, and *GRASS* grids can be imported into *WMS* and used for watershed modeling. The *File Import* command is used to specify which format and a file name for your DEM data. See the description of importing DEMs on page 2-15.

4.5 DEM Point Status

A DEM Point can have one of three different status': Active, Inactive, or NO DATA. An active DEM point is one with a valid elevation and is fully functional. An inactive DEM point is one with a valid elevation but is temporarily disabled. While disabled it will not be used for contouring, shading, or other functions involving the use of DEM points. A NO DATA DEM point is within the rectangular bounds of the DEM, but does not contain a valid elevation.

A series of commands can be used to set the status of the DEM points.

4.5.1 Set Active Region

The *Set Active Region* command is used to activate the currently selected DEM points and inactivate all DEM points not in the selected rectangle. You should use this command whenever you wish to isolate a sub-region of a DEM for watershed analysis.

4.5.2 Activate Selected

Activates all DEM points within the selected rectangle. Of course there has to be valid elevation in order for the DEM point to become active, i.e. you can not activate a NO DATA DEM point.

4.5.3 Activate All

Activate all DEM points with valid elevations.

4.5.4 Inactivate Selected

Inactivate DEM points within the selected rectangle. DEM points will become inactive but can still be viewed and later selected and reactivated.

4.5.5 Inactivate All

Inactivate all DEM points.

4.5.6 Delete Inactive

Inactive regions of the DEM can be eliminated altogether with this command. The main reason for deleting inactive cells is to reduce the size of the rectangular area of the DEM to free up memory resources for other operations. Since a DEM region must always be rectangular, “deleted” DEM points within the bounding rectangular will take on a NO DATA status.

4.6 Smoothing DEMs

In order to conserve the amount of disk spaced required to store a DEM, elevations are rounded to the nearest integer value. This causes elevation changes to occur in discrete steps rather than smoothly, as would be the case in nature. In regions of low relief, rounded elevations can cause an area to be artificially "flat." These flat areas, and all elevations can be restored to their natural smoothness by using the smoothing options found in the *DEMs* menu. When the *Smooth DEM* command is issued each elevation is smoothed according to the options specified in the *Smoothing Options* dialog.

GIS software such as *ARC/INFO*® and *GRASS* contain options for DEM smoothing as well. A grid file smoothed by one of these systems can be imported into *WMS*, eliminating the need to use the DEM smoothing utilities.

4.6.1 Smoothing Options

Different smoothing options can be set to specify how the smoothing process operates. Figure 4-2 shows the *Smoothing Options* dialog and a description of the different options follows.

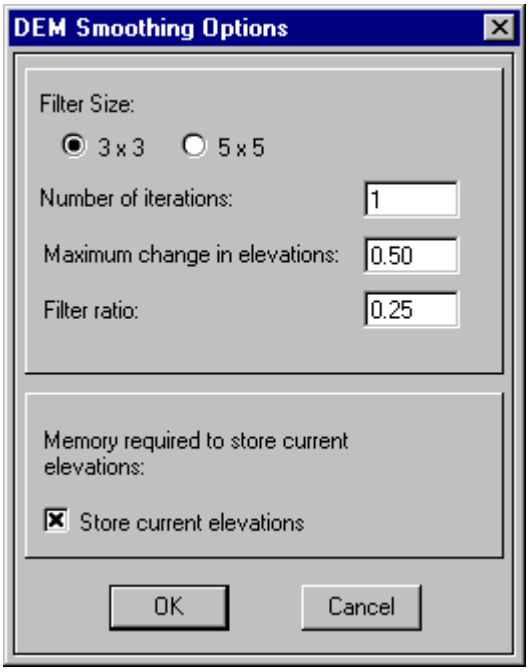


Figure 4-2 DEM Smoothing Options Dialog.

Filter Size

When a DEM is a smoothed an NxN filter matrix is placed over each elevation point and a new elevation is computed by taking an inverse-distance weighted average of all elevations within the filter (The weight assigned to the central cell is determined from the filter ratio). The dimension of N can be specified as either 3 or 5, meaning that new elevations are computed from either the nearest 8 or 24 neighboring points. When computing new elevations for points near the boundary, the number of neighboring points is modified to include only those portions of the filter which overlap the DEM.

Iterations

The number of smoothing iterations can be specified in the *Smoothing Options* dialog. By default only one iteration is done, but sometimes several smoothing iterations are required to propagate a change in elevations across a large flat area (If all neighboring points have the same elevation, no change will be made during the smoothing iteration).

Maximum Change in Elevation

A maximum change in elevation can be specified to insure that the integrity of the original DEM elevations is maintained. For example, if DEM elevations are rounded to the nearest meter, then smoothing should not adjust the elevation by more than plus or minus one half meter. This value can be increased, but care should be used in doing so in order to keep from "over smoothing" the original data.

Filter Ratio

The filter ratio should be between 0-1, and is used to specify the weight of the central cell of the filtering matrix. It can be used to establish how much effect the DEM point itself has on a newly computed elevation, and how much effect the neighboring cells have. For example, if the filter ratio is set to .75, then 75% of the newly computed elevation will be based on the point itself and 25% will be based on the neighboring elevation points.

Saving Current Elevations

Since the smoothing process can be iterative, and the best set of options for smoothing one DEM may be different than another, an option for storing original elevations exists. The *Restore Elevations* command in the *DEMs* menu can be used to set elevations back once they have been smoothed so that different options may be tried. Since saving elevations requires a significant amount of memory (4 bytes * number of DEM points), it may be necessary to turn this option off. If original elevations are not saved they can be restored by re-reading the DEM file.

4.6.2 Restoring Elevations

If current elevations have been saved, they may be restored at any time using the *Restore Elevations* command. This command can be used in conjunction with the different smoothing options in order to get the best possible result.

4.6.3 Removing Original Elevations

Since saving original elevations so that they can be restored requires a large amount of memory, they should be deleted once smoothing is done. Deleting this array of elevations is done using the *Remove Original Elevations* command.

4.6.4 Interpolating DEM Elevations to TIN

The *Interpolate to TIN* command is used to interpolate the elevations of the DEM to an existing TIN. If TIN vertices lie outside the bounds of the active region of the DEM no interpolation is performed. This interpolation is done automatically when creating a TIN from feature objects and a DEM is used for a background elevation map. If you are trying to create a TIN from a very large DEM (large number of DEM points) it may be advantageous to create the TIN first and then interpolate elevations from the DEM in a piece-wise fashion by dividing the DEM into several smaller regions and interpolating to the TIN one at a time.

4.6.5 Filling in Gaps Between DEMs

When reading in multiple DEM files that are adjacent, a small area or gap between the DEMs will have no elevation data. The Fill NODATA Cells command will interpolate an elevation for the selected DEM cells that are classified as "NODATA." The elevation for a selected NODATA cell is determined using inverse distance weighted interpolation from it's eight nearest neighbor cells (if any of the eight neighboring cells are NODATA cells then they are not used in the interpolation). This command is intended to correct single isolated DEM points or a single row/column, and is not intended to create data for large regions of NODATA cells, especially regions on the border of the DEM.

4.6.6 Lat Lon Calculator

The *Lat Lon->UTM* command can be used to convert a pair of latitude-longitude coordinates into their corresponding UTM coordinates. This is useful when working with DEM data or gage locations in order to properly geo-reference objects with data being worked on in *WMS*. Coordinates can be entered in either the degrees-minutes-seconds format or as seconds only as show in Figure 4-33. To locate the UTM coordinates calculated in this dialog in your data, it may be useful to toggle on the *Create feature point at this location* check-box. This will create a point in the *Graphics* window at the given UTM coordinates.

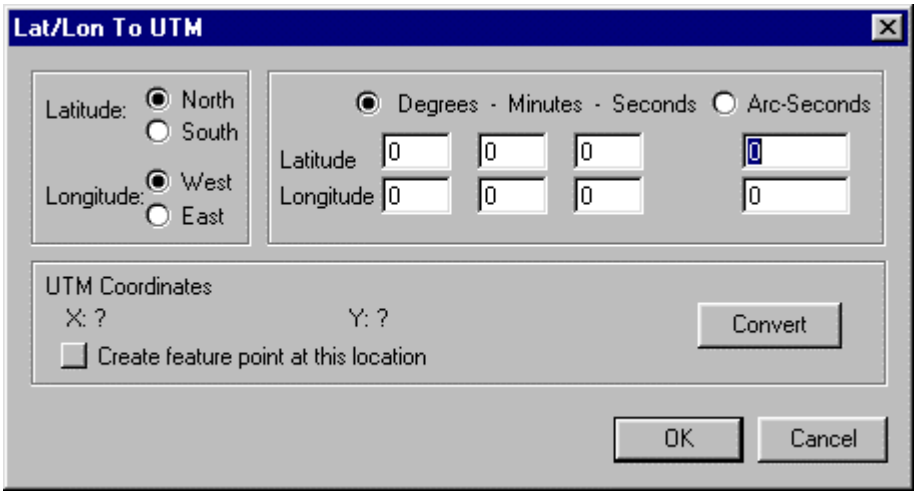


Figure 4-3 Lat Lon to UTM Dialog.

CHAPTER 5

Drainage - DEMs

5.1 Introduction

The previous chapter discussed all of the tools used to manipulate, smooth, and visualize DEMs. A DEM can further be used to provide a background elevation source for the creation of a TIN from feature objects, or to perform drainage analysis using information derived from the elevation points. Data, such as flow directions, flow accumulations, and basin id's can be computed and stored as "attributes" of the DEM at each valid DEM point.

When determining stream networks and watershed boundaries, the DEM works in combination with the *Map* Module. Connected DEM points that comprise a stream branch are converted to arcs (see the definition of watersheds in the Map Module chapter), and groups of DEM points that make a sub-basin are converted to polygons for further hydrologic model definition.

Besides the elevation DEM, a flow direction for each elevation point in the DEM is required in order to perform drainage analysis. Elevation and flow direction are the essential data from which all of the other drainage computations are made. Flow directions can be computed with *ARC/INFO*® or *ArcView*® Spatial Analyst, GRASS, or TOPAZ. A special version of TOPAZ has been created for use with *WMS* that only requires an elevation grid as input and produces a flow direction grid as output.

This chapter describes all of the tools and functions available for delineating stream networks and basin boundaries, and development of hydrologic data from these elevation DEMs and flow directions.

5.1.1 Limitations and Memory Considerations

Developing watershed data with DEMs in *WMS* can be very memory intensive and there are certain limits. While *WMS* and TOPAZ (the program used to develop flow direction data from DEMs) both use dynamic memory allocation, the memory and time required for computation/display can become prohibitive. Be sure to provide time for the individual processes to be performed. Also you will likely need more than 64 (or higher) Megabytes of RAM and plenty of swap space to do problems in this size range.

5.2 Tool Palette

Besides the select DEM point and contour label tools described in the previous chapter, the following tools have been added to the DEM module to aid in drainage analysis. Because hydrologic data development with DEMs requires tight integration with the *Map Module*, many of these tools are copies of the same tools used in that module. Refer to the *Map Module* chapter for a complete description of these tools, in this chapter only the uses for the tools as they relate to performing drainage analysis with a DEM is discussed.

5.2.1 Flow Path

Draws a flow path from a selected point on the DEM. This options only works if a flow direction grid has been imported to accompany the DEM.

5.2.2 Select Points/Nodes

The *Select Points/Nodes* tool is used to select existing points or nodes so that they can be converted to outlet points. Double clicking on a point or node with this tool brings up the *Point* or *Node Attribute* dialog.

5.2.3 Select Vertex

The *Select Arc Vertices* tool is used to select vertices on an arc.

5.2.4 Select Arc

The *Select Arc* tool is used to select arcs for operations such as deletion, or changing attributes. Double clicking on an arc with this tool brings up the *Arc Attribute* dialog.

5.2.5 Create Points

The *Create Points* tool is used to interactively create new points using the cursor. These new points could be defined at outlet locations for the DEM.

5.2.6 Create Arc Vertex

The *Create Arc Vertex* tool is used to interactively create new vertices along an existing arc, to add more detail.

5.2.7 Create Arc

The *Create Arc* tool is used to interactively create new arcs. An arc is created by clicking once on the location where the arc is to begin, clicking once to define the location of each of the vertices in the interior of the arc, and double-clicking at the location of the end node of the arc. Arcs are used to define the channel network for a watershed or to add detail to an existing one.

When creating stream arcs, the points/vertices must be connected from downstream to upstream as the arc is entered. Stream arcs are drawn with an arrow pointing in the downstream direction and can be reversed after creation using the *Reverse Directions* command.

5.2.8 Select Polygon

The *Select Polygon* tool is used to select previously created sub-basin polygons. Double-clicking on a polygon with this tool brings up the *Polygon Attributes* dialog and allows you to change the type of polygon to a basin polygon.

5.2.9 Select Branch

The *Select Branch* tool can be used to automatically select all arcs of a branch without having to select each individually. This is particularly useful when locally redistributing vertices along a stream branch. A branch is selected by selecting any arc in the branch. *WMS* searches down the stream from the selected arc until the next branching node is encountered, and then adds all arcs upstream from that node to the list of selected arcs.

5.2.10 Select Network

The *Select Network* tool can be used to select all arcs of a given network. It works similarly to the *Select Branch* tool except that when an arc is selected

WMS searches downstream until a node with no downstream arcs is encountered instead of stopping at the next downstream branching node.

5.2.11 Place Drainage Label

The place drainage label tool can be used to reposition the location of where the drainage attributes for a basin are placed. The default location to display area, slope, and other computed drainage parameters is at the centroid of the basin. However, the display often becomes cluttered and it may be necessary to move the display of labels to a different location. If the label is moved a long distance then an arrow pointing to the basin containing the parameters is created. The first location you click on in a basin, when this tool is active, becomes the position the arrow points to. While holding down the left mouse key you can then drag the label to a new location and let up. The label will then be centered around this last point.

5.3 Display Options

The display options control which DEM point attributes are displayed. Each display feature associated with drainage for DEMs is listed in the *DEM Drainage Display Options* dialog (Figure 5-1) accessed by selecting the *Display Options* command in the *Drainage* menu. The check box next to the feature name can be toggled on or off to control whether or not the feature is to be displayed. In addition, the color button to the left of the check box can be used to set the color and other appropriate attributes such as radius, line thickness, line style etc. General purpose DEM display options were documented in the previous chapter and the display options relating to drainage delineation are documented here.

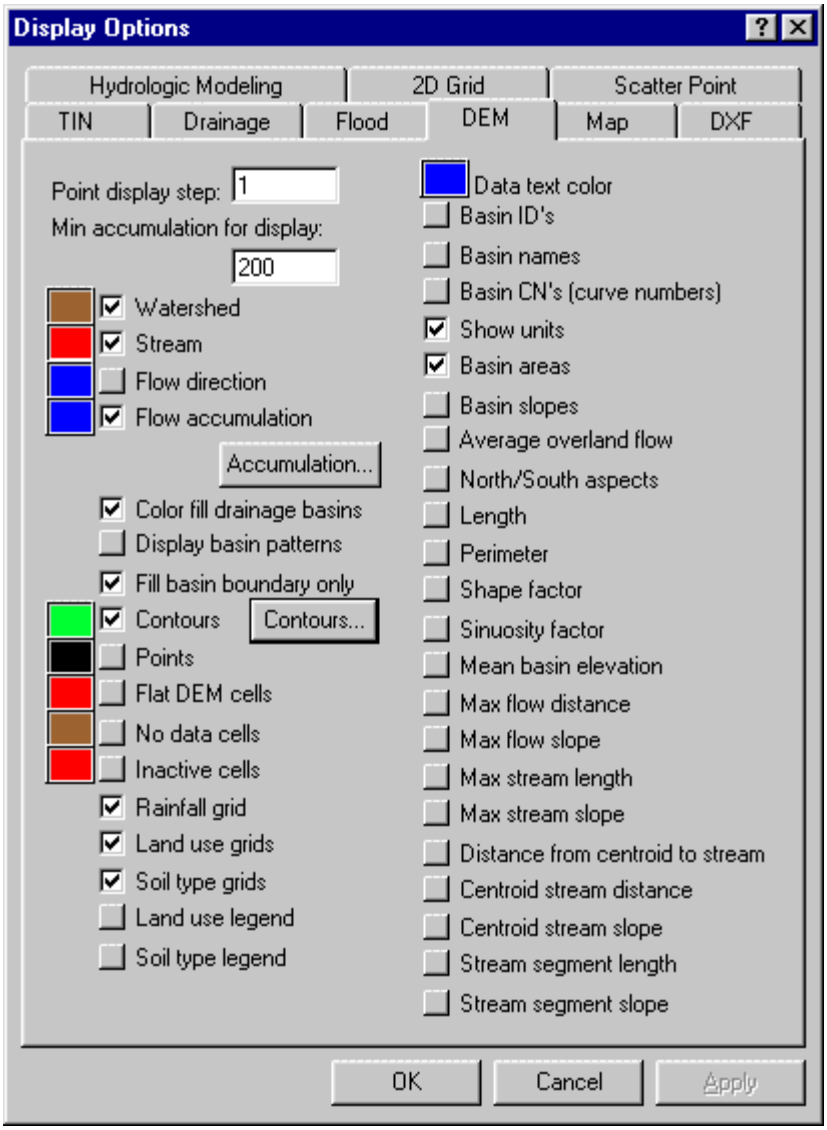


Figure 5-1 DEM Drainage Display Options Dialog.

5.3.1 Watershed

Any cell which belongs to a sub-basin is displayed in the chosen color. This is different than filling drainage basins in that all sub-basins are given the watershed color.

5.3.2 Stream

Any cell beneath a stream arc is displayed in the chosen color when this option is toggled on.

5.3.3 Flow Directions

A small arrow from the DEM point to the neighboring cell in which flow is defined is displayed. This option only works when a flow direction grid has been computed or imported. This option is most effective when zooming in since the arrows become so small for most DEMs when the entire region is displayed.

5.3.4 Flow Accumulations

After a flow accumulation grid has been created or imported, DEM points whose accumulation (number of DEM points whose flow path passes through the given point) is greater than a defined threshold are displayed. The *Accumulation Opts.* dialog allows you to set different colors for different threshold values.

5.3.5 Color Filled Drainage Basins

When toggled on, each basin is displayed with a unique color so that drainage boundaries are easily identified.

5.3.6 Display Basin Patterns

WMS cycles between colors to display basin boundaries. If a large number of basins are used then two may have the same color. To avoid this problem you can turn on the *Display basin patterns* option so that in addition to cycling through colors, the fill pattern is also changed.

5.3.7 Fill basin boundary only

Filling basins with a color can be very time consuming. In order to speed up the display you can turn on the *Fill basin boundary only* option so that only the DEM cells on the boundary of the basin are colored.

5.3.8 Basin Data Attributes

All of the display options listed in the right column represent computed watershed modeling parameters. They can be turned on or off for display once a watershed has been delineated and basin data parameters are computed. The text for all basin data is the same and can be modified selecting the *Data text color* button.

5.4 Flow Directions & Accumulations

A flow direction grid consists of a flow direction value for each DEM point The flow direction identifies which neighboring point has the lowest elevation. A flow accumulation grid consists of an integer value for each DEM point that represents the number of "upstream" DEM points whose flow path passes through it. High accumulation values indicate points in the stream, whereas low values represent areas of overland flow.

Flow directions and accumulations are typically determined using a program, such as *ArcView*®, GRASS, or TOPAZ. Resulting grid files can then be imported into *WMS*. These programs all use variations of the eight-point pour model (Puecker and Douglas, 1975, and Garbrecht, 1995). Figure 5-2 illustrates how flow directions are computed by determining which of the eight neighboring DEM points has the lowest elevation. The flow direction value for that DEM point is then assigned an integer number representing the given direction.

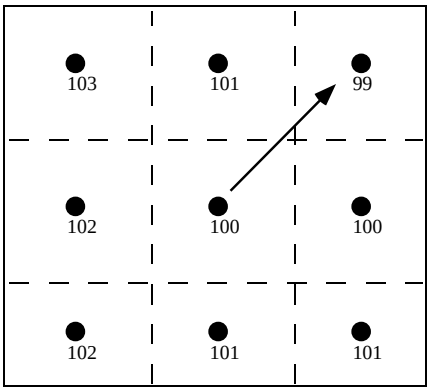


Figure 5-2 Eight-Point Pour Model.

If all DEM points had one and only one lower neighbor, the process of determining flow directions would be simple and the requirement to use other programs would not exist. However, there are many subtle problems dealing with depressions and flat areas which make the algorithm for determining flow directions complex. Computation of flow accumulations are fairly straight-forward once the flow directions are determined.

At this point computations of flow directions can not be done directly by *WMS*. However, a version of the TOPAZ (see section 0) program, modified specifically to work with *WMS*, creates as output the flow direction and flow accumulation grids. These grids can then be imported as DEM attributes and used for basin delineation. Figure 5-3 shows the dialog used to launch TOPAZ for computing flow direction and flow accumulation grids.

The result of this command is that *WMS* will create two TOPAZ input files: DEDNM.INP (DEM elevations used by TOPAZ), and DNMCNT.INP (a

control file). After creating these two files *WMS* starts the TOPAZ module DEDNM which produces as output a pre-processed elevation file (RELIEF.DAT), a flow direction file (FLOVEC.DAT), a flow accumulation file (UPAREA.DAT), and a report file (DEDNM.RPT). You can choose which files DEDNM saves as output, but if you wish to perform basin delineation with DEMs in *WMS* then as a minimum you must choose the flow direction file.

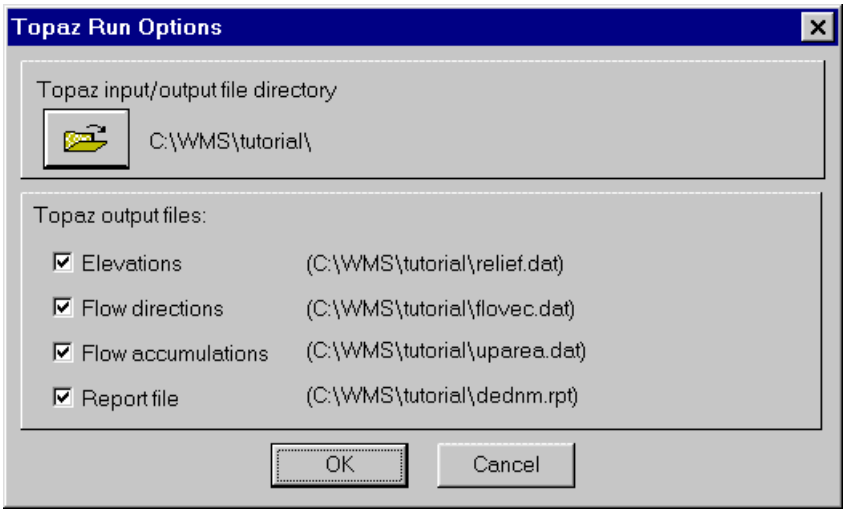


Figure 5-3 Flow Directions Dialog.

Besides TOPAZ, flow directions (and accumulations) may also be computed by *ARC/INFO*®, *ArcView*®, or GRASS. No matter which program you use to create flow directions, you will need to import them using the *Flow Directions* command. The *Import Flow Directions* dialog (see Figure 5-4) allows you to specify the file format of the flow direction grid and then prompts you for the file name.

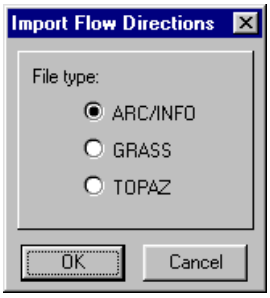


Figure 5-4 Import Flow Directions Dialog.

Once flow directions have been imported into *WMS*, flow accumulations can either be computed or imported (if you use TOPAZ to compute flow directions you will have already computed the flow accumulations as well and they will be in a file named UPAREA.DAT). Flow accumulations are computed by counting, for each DEM point, the number of DEM points whose flow paths

pass through the DEM point. Streams will be identified by large accumulation values since the flow paths of many points pass through the stream points. For example the outlet of a watershed should have the highest flow accumulation of any of the DEM points since the flow paths all points in the watershed will eventually pass through the outlet point. Flow accumulations may also be imported as a result of using one of the other GIS programs previously mentioned. The *Flow Accumulations* dialog is shown in Figure 5-5.

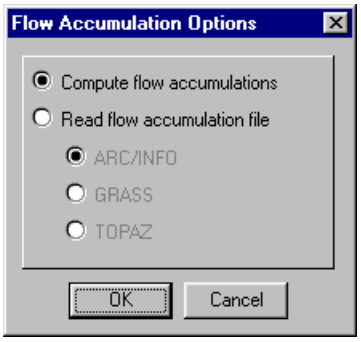


Figure 5-5 Flow Accumulations Dialog.

Even though *WMS* can compute flow accumulations, it may be more efficient to import a flow accumulation grid from TOPAZ, *ARC/INFO*®, or GRASS.

5.5 Drainage Analysis

As stated in the introduction, the elevation and flow direction values for each DEM point are the primary data required for performing basin delineation and watershed characterization with DEMs. Once these data are imported and flow accumulations computed, stream networks and basin boundaries are defined with the aid of feature objects. Arcs representing streams and feature points or nodes representing basin outlets must be present in order to define basins. Once basins are defined, watershed and sub-basin boundaries can be converted to feature polygons. All of the ties to the hydrologic models are made available through these feature objects with geometric values such as area, slopes, lengths, etc. being populated from the DEM data.

5.5.1 DEM Streams->Feature Arcs

The *DEM->Stream Arcs* command is used to create feature arcs from DEM points whose flow accumulation values are above a defined threshold. An arc vertex is created for each DEM point that has a flow accumulation value greater than the threshold entered. Consecutive stream DEM points are then joined together as arcs with nodes created at junction points where the stream splits.

By default stream arcs are created for all DEM points that have a flow accumulation larger than the threshold. Outlet feature points/nodes are created at DEM points which pass the accumulation threshold and do not have a neighboring point with a higher accumulation. The stream is "traced" upstream by noting the neighboring DEM point with the next highest accumulation. This process is repeated until no neighboring point has an accumulation larger than the threshold.

As an alternative to automatically creating outlet points, a set of outlet feature points can be created manually using the *Create Points* tool prior to using the *DEM->Stream Arcs* command. The *Stream Feature Arcs Opts* dialog shown in Figure 5-6 can then be used to specify that streams will be created from the pre-defined outlets only.

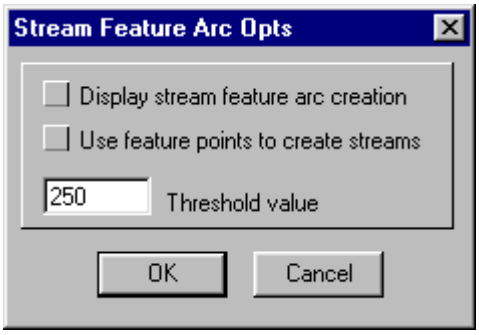


Figure 5-6 Stream Feature Arc Opts Dialog.

Outlet points can be created at any DEM point, but should be in a DEM point that has a high enough flow accumulation to pass the threshold. The Flow Accumulations display option can be very useful for identifying these points and for determining what an appropriate threshold value is.

The *DEM Streams->Feature Arcs* command can also be very useful for defining stream arcs which are later used for creating a TIN surface.

5.5.2 Defining Basins

Each time a feature outlet point is created, a basin for each upstream feature arc is created for the hydrologic modeling tree. This means that the stream arcs themselves are associated with a basin even before the *Define Basins* command is issued. When the command is used the DEM points intersected by the stream arcs are assigned the basin id already given to the arcs. The *Define Basins* procedure then continues by tracing the flow paths of the remaining DEM points until a point which has already been assigned a basin id is intersected. The result is that each DEM point gets assigned the id of the sub-basin it belongs to.

Additional outlet points can be created by changing the attribute of existing arc nodes to outlets or by converting arc vertices to nodes and then changing the attribute to outlet. The *Node<->Outlet* command can be used to accomplish this. Any selected node or vertex will automatically be converted to an outlet node when using this command. Any selected outlet node will automatically be converted to a generic node type when using this command.

The *Define Basins* command can be used any number of times to redefine basins after the addition/deletion of outlet nodes.

5.5.3 Inactivate Null Basin

The *Inactivate Null Basin* command is used to inactivate all DEM points which are not currently assigned a basin id. The inactive DEM points can then be eliminated altogether (freeing up valuable memory resources) by choosing the *Delete Inactive* command from the DEMs menu.

5.5.4 DEM Basins Boundaries to Polygons

Once the desired sub-basin delineation from the DEM points has been defined, the basin boundaries can be converted to feature polygons. This is done by tracing the boundaries between sub-basins to generate arcs. After all of the boundaries have been defined the arcs are converted to polygons and the polygons assigned the appropriate basin id.

5.5.5 Drawing Flow Patterns

The *Draw Flow Patterns* command initiates a flow path from the DEM points according to the current display step. By drawing flow paths from the DEM points a could visual queue of the watershed flow patterns can be obtained. If a basin polygon is selected prior to issuing the command then flow paths will only be drawn for the DEM points that are part of the selected basin. The display step of the flow patterns can be controlled by modifying the *Point Display* step option in the *DEM Display Options* dialog.

5.5.6 Computing Basin Data

After defining basin boundaries, attributes such as basin areas and slopes and stream lengths and slopes can be computed using the *Compute Basin Data* command. These are all geometric parameters used in defining basins and routing networks in *HEC-1, TR-20,* and other hydrologic models. If the basins are changed in any way, the drainage data must be recomputed using this command. When computing basin data the model units and the parameter units must be specified. The only choices available for model units are feet and meters whereas the parameter for area include square miles, square kilometers and acres, and for distance include mile, kilometer, feet, and

meters. A complete definition of the different geometric attributes computed and how they may be used to compute travel times (lag time, time of concentration) is given in the *Hydrologic/Hydraulic Calculations* chapter (see section 15.3.1).

5.5.7 Merge Selected Basins

The Merge Selected Basins command merges two selected sub basins into a single basin. The two sub basins must be directly connected to the same outlet point.

5.5.8 Polygon Basin ID's -> DEM

The *Polygon Basin ID's->DEM* command assigns Basin ID's to the DEM from a set of polygons that represent basin boundaries. This command is useful if you have not delineated your basin using the flow directions and flow accumulations from within *WMS*, but instead have a set of polygons representing basin boundaries with unique ID's. Once basin ID's have been assigned then basin data such as area, average slope, etc. can be computed.

You would likely only use this option if you had a set of feature objects already and/or wish to "over-ride" the basin boundaries that are determined from elevation data. This might occur in an urban watershed where streets, canals, etc. may not be apparent in the digital elevation data.

5.6 TOPAZ

The TOpagraphic PArameteriZation program (TOPAZ) was developed by the USDA-ARS, National Agricultural Water Quality Laboratory under the direction of Dr. Jurgen Garbrecht. TOPAZ is a public domain program that is distributed free of charge to interested persons. A modified version of the program is distributed with *WMS* for the purpose of computing flow directions for use in basin delineation with DEMs directions (see section 5.3.3). However, TOPAZ is capable of further DEM elevation processing, including raster smoothing, flow accumulation computations, basin and stream delineation and ordering, and development of other watershed parameters. If you are interested in obtaining the latest, complete version of TOPAZ you should write Dr. Jurgen Garbrecht at the following address, providing your name, address, phone, fax, and email address:

Dr. Jurgen Garbrecht
Grazinglands Research
7207 West Cheyenne St.
El Reno, OK 73036

If you have or obtain the complete TOPAZ program, *WMS* is capable of writing an input file for DEDNM (the primary TOPAZ module). DEDNM requires as input a file containing the elevations (must be named DEDNM.INP and a control file named DNMCNT.INP. Figure 5-7 shows the DNMCNT options dialog, and the following paragraphs outline what the different options are capable of.

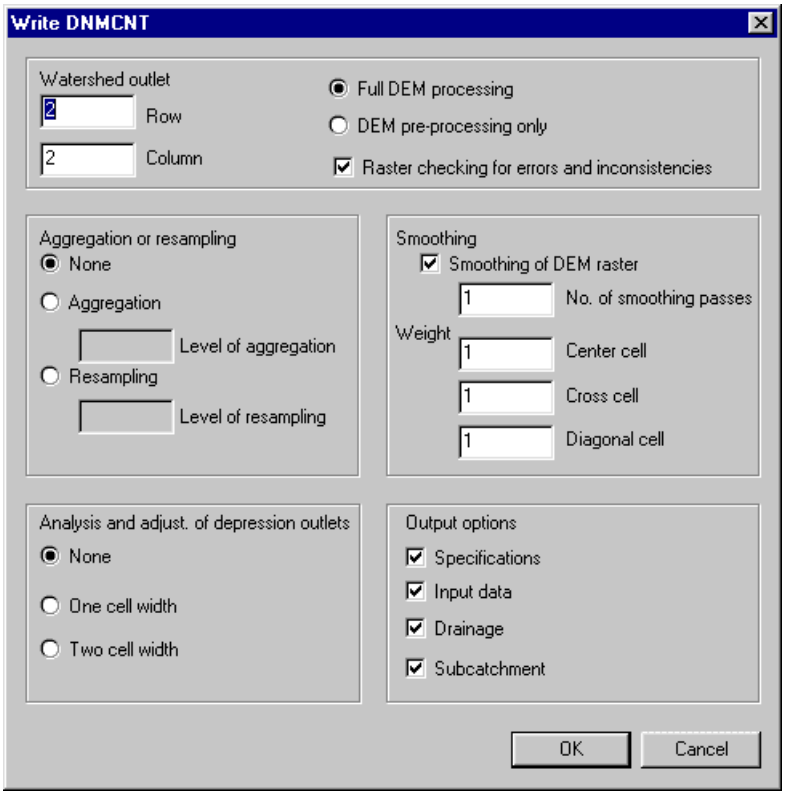


Figure 5-7 DNMCNT Options dialog.

5.6.1 DEM Processing

Full DEM processing includes flow direction/accumulation computations and watershed delineation, whereas the pre-processing only option will only aggregate and smooth the DEM resulting in a new elevation DEM.

5.6.2 Watershed Outlet

The watershed outlet is the DEM row and column where the watershed outlet is. You can change this during run time to evaluate different locations.

5.6.3 Aggregation/Resampling

Aggregation and resampling result in a reduction of data since DEM points in an nXn window are reduced to a single point. The level of aggregation/resampling can be from 2 to 20. The result is an elevation DEM with reduced resolution. The difference between the two options is that resampling interpolates to find the new elevation value for each new DEM point whereas aggregation simply takes the elevation point nearest the "center" of the moving window.

5.6.4 Smoothing

The TOPAZ smoothing option is similar to the one performed internally by *WMS* (see section 4.6). An 3X3 window is passed over the DEM and a new elevation value is computed based on interpolation of the 9 DEM points in the window. Weighting factors can be assigned to the center, cross, and diagonal cells.

5.6.5 Analysis and Adjustment of Outlet Depressions

When removing interior pits and flat depressions it is often necessary to adjust some DEM points in the area where a depression needs to breach a higher DEM point to "flow" into a region of lower elevations. This option determines how many DEM points can be adjusted in order to create a breach from a depression to lower elevation DEM point.

5.6.6 Output Options

Specifies which files will be saved as a result of the DEDNM execution

CHAPTER 6

TINs

6.1 Introduction

One of the basic modules in *WMS* is the *Triangulated Irregular Network (TIN)* module. The tools and commands in the *TIN* module provide the basic operations for terrain modeling and basin delineation. TINs are formed by connecting a set of xyz points (scattered or gridded) with edges to form a network of triangles. The surface is assumed to vary in a linear fashion across each triangle.

6.2 Tool Palette

The following tools are in the dynamic portion of the *Tool Palette*. These tools are available when the *TIN* module is activated. Only one tool is active at any given time. The action that takes place when the user clicks in the *Graphics Window* with the cursor depends on the current tool. The tools are for selection and interactive editing of TINs.

6.2.1 Select Vertices

The *Select Vertices* tool is used to select vertices for operations such as deletion, or to drag a vertex to a new location. The coordinates of selected vertices can also be edited using the *Edit Window*. This same tool allows for selection of outlet points on the TIN.

6.2.2 Select Triangles

The *Select Triangles* tool is used to select triangles for operations such as deletion. In addition to the standard multi-selection options, another type of multi-selection is available with this tool. By holding down the *CTRL* key while dragging the cursor, a selection line can be entered. All triangles intersected by the line are added to the selection list.

6.2.3 Select TIN

The *Select TIN* tool is used to select one of the TINs currently in use by *WMS*. When this tool is active a triangular icon is placed at the centroid of the TIN so that it can be selected by clicking in the icon. Selected TINs can be used to set the active status or to merge with another TIN.

6.2.4 Select Vertex Strings

The *Select Vertex Strings* tool is used to select one or more strings of vertices. Vertex strings are used for operations such as adding breaklines to the TIN or selecting a string of vertices which will be used to create a stream.

The procedure for selecting vertex strings is somewhat different than the normal selection procedure. Strings are selected as follows:

- Click on the starting vertex for the string. The vertex selected will be highlighted in red.
- Click on any subsequent vertices you would like to be part of the string (vertices do not have to be next to each other) and double click on the final vertex. The vertices selected are now connected by a solid red line.
- To remove the last vertex from a string, press the *BACKSPACE* key. To abort entering a vertex string, press the *ESC* key. To end a vertex string, press *RETURN* or double click on the last vertex in the string. Another vertex string can then be selected.

6.2.5 Create Vertices

The *Create Vertices* tool is used to manually add vertices to a TIN. It can only be used in plan view. When this tool is selected, clicking on a point within the *Graphics Window* will place a new vertex at that point. What happens to the vertex after it is added (whether and how it is triangulated into the TIN) depends on the settings in the *Vertex Options* dialog under the *TINs* menu.

These settings can easily be used to digitize elevation data from scanned images of contour data as described in section 6.5.

6.2.6 Create Triangles

The *Create Triangles* tool is used to manually create new triangles. Triangles are normally created by triangulating a set of points automatically. However, this tool is useful for localized editing and refining a TIN. To use the *Create Triangles* tool you may either:

- Select three vertices that will form the triangle. The vertices can be selected in either clockwise or counter-clockwise order.
- Drag a box around three vertices that will form the triangle.

The *ESC* key can be used to abort the creation of a triangle once you have started selecting vertices.

6.2.7 Swap Edges

The *Swap Edges* tool swaps the edges of two adjacent triangles and is usually used to make local adjustments to a TIN. To use the tool, simply click on any triangle edge.

6.2.8 Contour Labels

The *Contour Label* tool manually places numerical contour elevation labels at points clicked on with the mouse. These labels remain on the screen until the contouring options are changed, until they are deleted using the *Contour Label Options* dialog, or until the *Graphics Window* is refreshed. Contour labels can also be deleted with this tool by holding down the *SHIFT* key while clicking on the labels. This tool can only be used when the TIN is in plan view.

6.2.9 Select Basins

The *Select Drainage Basins* tool is used to select basins which can then be either merged together or split. In addition to selecting basins from the TIN, this tool can be used to select one of the basin icons.

6.2.10 Place Basin Labels

Computed drainage data may be displayed for each basin. However, when there are many basins, the screen can become cluttered with data. The *Place Basin Labels* tool allows basin data to be placed at a position other than the

centroid, which is the default location. When moving a label, click in the desired basin and while holding down the mouse button, drag the cursor to the desired position on the screen and then release the button. An arrow will be drawn from the final position to the point first clicked in the basin.

6.2.11 Flow Path

The *Flow Path* tool allows the flow paths for specified points to be drawn. When this tool is active, clicking in the graphics window at a location on a TIN will cause a flow path to be initiated from that point and followed "downstream" until a pit or local minima is reached, or until the path leaves the TIN. Overland flow and Stream flow are distinguished by setting the colors for Downhill Overland and Downhill Channel in the *Drainage Display Options* dialog. By default, light blue specifies overland flow (across triangle faces), and dark blue specifies channel flow (along channel edges). This tool can be very useful in checking portions of an edited TIN before stream and basin definition is completed.

The length and slope of overland and stream flow is displayed in the help window each time a new path is drawn. This can be helpful in obtaining parameters used to compute lag times with some empirical formulas. Stream distances are shown only after a stream has been created. In other words, channel flow is not counted in the stream distance unless a "stream" has been created along the channel.

Flow paths initiated from the centroid of each triangle can be displayed using the *Draw Flow Patterns* command in the *Drainage* menu.

6.3 Display Options

Display options control which features of the TIN are displayed. Each display feature associated with TINs is listed in the *TIN* tab of the *Display Options* dialog (Figure 6-1) accessed by selecting the *Display Options* command in the *Display* menu. The *TIN* tab will be displayed on top whenever you access the display options from the *TINs* module. The check box next to the feature name can be toggled on or off to control whether or not the feature is to be displayed. In addition, the color button to the left of the check box can be used to set the color and other appropriate attributes such as radius, line thickness, line style etc.

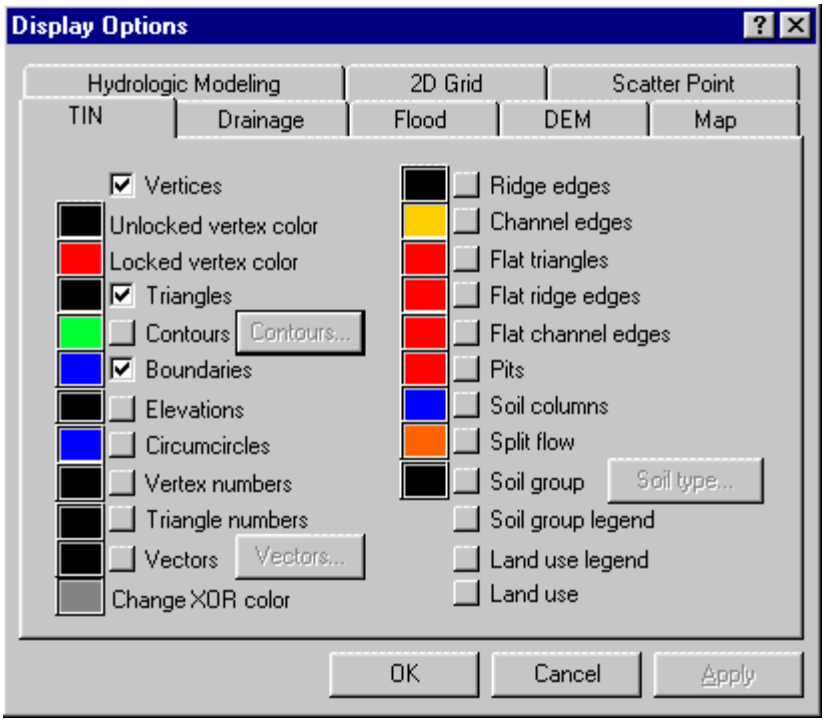


Figure 6-1 TIN Display Options Dialog.

6.3.1 Vertices

If the *Vertices* item in the *TIN Display Options* dialog is set, the TIN vertices will be displayed each time the *Graphics Window* is refreshed. Both a "Locked" and "Unlocked" vertex color may be set so that there is a visible difference when displaying the TIN.

6.3.2 Triangles

If the *Triangles* item in the *TIN Display Options* dialog is set, TIN triangles will be displayed each time the display is refreshed.

6.3.3 Contours

If the *Contours* option in the *Display Options* dialog is set, the TIN is contoured when the display is refreshed. The color corresponds to the default contour color. However different coloring schemes may be chosen using the *Contour Options* dialog accessed from within the *Display* menu.

6.3.4 Boundaries

If the *TIN Boundary* item in the *TIN Display Options* dialog is set, the boundary of the TIN will be displayed each time the *Graphics Window* is

refreshed. This feature is often used in conjunction with contours in order to display the TIN boundary without cluttering the screen by displaying each triangle.

6.3.5 Circumcircles

If the *Circumcircles* item in the *TIN Display Options* dialog is set, the circumcircle enclosing the three vertices for each triangle will be drawn when the display is refreshed. Circumcircles provide the basis of a Delauney triangulation since the Delauney criterion is satisfied by ensuring that no circumcircle encloses a vertex. Displaying circumcircles can aid in the understanding of the triangulation process.

6.3.6 Vertex/Triangle Numbers

Displays the ID's of the triangles and triangle vertices.

6.3.7 Vectors

If a data set (used primarily with the two-dimensional runoff models such as CASC2D) has vector data associated with it then the display of a vector representing direction for each element can be toggled on. The *Vector Opts…* button allows you to adjust the display quality/options of the vectors.

6.3.8 Elevations

If the *Vertex Elevations* item in the *TIN Display Options* dialog is set, the elevation of each vertex is displayed adjacent to the vertex.

6.3.9 Ridge Edges

If the *Ridge Edges* option is set, all edges where both adjacent triangles slope away from each other are displayed.

6.3.10 Channel Edges

If the *Channel Edges* option is set, all edges where both adjacent triangles slope towards each other are displayed.

6.3.11 Flat Triangles

If the *Flat Triangles* option in the *Display Options* dialog is set, all flat triangles will be displayed on the *Graphics Window*. Flat triangles occur where three vertices with the same elevation form a triangle. Displaying them can

aid in editing TINs so that stream networks and drainage basins can be delineated.

6.3.12 Flat Ridge Edges

If the *Flat Ridge Edges* option is set, all flat ridge edges will be displayed. Flat ridge edges occur where two adjacent triangles slope away from each other and where the Z elevation of the two vertices comprising the common edge are the same. Flat ridges must be corrected before refinement of drainage boundaries can be done.

6.3.13 Flat Channel Edges

If the *Flat Channel Edges* option is set, all flat channel edges will be displayed. Flat channel edges occur where two adjacent triangles slope towards each other and where the Z elevation of the two vertices comprising the common edge are the same. Flat channel edges need to be corrected before stream networks can be defined completely.

6.3.14 Pits

If the *Pits* option is set, all pits will be displayed. Pits correspond to local minima on a TIN where water would congregate. They make up a subset of the default outlets since flow terminates there.

6.3.15 Soil Columns

Display a soil column definition for an X-Model simulation.

6.3.16 Split Flow

Because of the way vertices are triangulated, it is possible to have splitting flow paths from some of the TIN vertices. Such vertices can create problems for the drainage delineation algorithms if the different flow paths end up in separate basins. Toggling this display option will cause all such vertices to be displayed so that they can be corrected. Determining which vertices are split flow vertices can be time consuming, so you will want to turn this option off once split flow vertices have been corrected.

6.3.17 Soil Group

If this option is set, a color fill pattern for the hydrologic soil group will be used to display the triangle. The *Soil Type Display Options* button allows you to toggle on/off the display of individual soil types. You may also change the color and/or pattern used for a given soil.

6.3.18 Land Use

If the *Land Use* option is set, all triangles of similar land use will be filled with a unique color/pattern. A land use table must be defined using the *Import* command from the *File* menu and selecting the *Land use/Soil type tables* option. Land use and soil types are assigned to TIN triangles when the *Compute Composite Runoff Coeff/CN* command is used.

6.3.19 Hydrologic Soil Group Legend

If the *Hydrologic Soil Group Legend* is turned on, a legend showing the pattern for each group will be displayed in the upper right corner of the *Graphics Window*.

6.3.20 Land Use Legend

If the *Land Use Legend* is turned on, and a table of land use data has been read, a legend showing the color and pattern for each defined land use will be displayed in the upper left corner of the *Graphics Window*. The flood legend and land use legend should not be turned on at the same time.

6.4 TIN Attributes

While *WMS* will allow you to load multiple TINs at the same time, only one TIN can be designated as the "active" or working TIN. The active TIN may be set by selecting the *Select TIN* tool, selecting the TIN you wish to make active, and then choosing the *Make Active* command in the *TINs* menu. It can also be set by double-clicking on the TIN when using the *Select TIN* tool.

The *Attributes* command found in the *TINs* menu can be used to set the name of the TIN. The name of the TIN is always displayed along with the TIN icon when using the *Select TIN* tool.

6.5 Vertex Options

6.5.1 Creating New Vertices

New vertices can be created by selecting the *Create Vertices* tool from the *Tool Palette* and clicking in the *Graphics Window* where the new vertex is to be located. The x and y values of the vertex are determined by the position of the mouse cursor when a click is made. The z value must be entered separately. A default z value and other parameters governing the creation of new vertices can be set by selecting the *Vertex Options* command from the *TINs* menu.

One of the easiest ways to create a TIN for a small area where a paper contour map exists is to use this tools and follow the steps outlined below:

1. Scan the paper map and save it as a TIFF (*.tif) image.

2. Register the image as described in section 3.4 (you may want to mark the map with your register points prior to scanning it).

3. Set the vertex options (see section 6.5.4 below) so that you are not prompted for a z-value each time and so that no triangulation takes place as you create new vertices (basically turn off all of the vertex options).

4. Set the default z-value to the contour value you wish to digitize first.

5. Digitize or create vertices along the specified contour value (the spacing of points along the contour lines should be approximately the same distance as the spacing between adjacent contours).

6. Repeat steps 4 and 5 for each contour line.

7. Triangulate the vertices once you are done.

Of course this method is awfully tedious for larger areas, but is ideally suited for smaller areas where there are not too many contours to be digitized.

6.5.2 Deleting Vertices

Vertices can be deleted by selecting the vertex(s) to be deleted and hitting the *DELETE* or *BACKSPACE* key on the keyboard or by selecting the *Delete* command from the *Edit* menu. If the *Confirm Deletions* option in the *Edit* menu is active, you will be prompted to confirm each deletion. This is helpful in preventing accidental deletions. The confirm-deletions flag can be toggled by selecting the *Confirm Deletions* item.

6.5.3 Changing Vertex Positions and Z Values

Two methods of editing vertex positions and z values are available. To manipulate vertex positions and z values, the *Select Vertex* tool must be selected.

- A vertex can be moved to a new position by clicking on the vertex and holding down the mouse button while dragging the vertex to the desired position.

- If the current view is plan view, dragging the vertex will cause it to move in the xy plane. *WMS* will not allow the vertex to be dragged to

a position where one of the surrounding triangles would become inverted.

- If the current view is not the plan view, the vertex will move along the z-axis.

- The vertex position and z value can also be manipulated by selecting the vertex and changing the xyz values that will appear in the x, y, and z edit boxes.

Display options such as contours are updated automatically as a vertex's position is altered as long as these options are selected from the *TIN Display Options* dialog.

6.5.4 Vertex Options Dialog

The *Vertex Options* dialog (accessed from the *TINs* menu) contains six options as shown in Figure 6-2:

- If the *Retriangulate after deleting* is checked, the region surrounding the vertex will be retriangulated as each vertex is deleted. Otherwise, the triangles adjacent to the vertex are simply deleted.

- If the check box entitled *Adjust boundary to include exterior vertices* is selected, the boundary of the TIN will be changed so that the new vertex becomes part of the TIN if a new point is added outside the active TIN. If the new vertex is in the interior of the active TIN, the vertex will be automatically incorporated into the TIN.

- The default z value edit box displays the z value that will be assigned all subsequent new vertices created with the *Create Vertex* tool if the *Confirm z values* check box is not selected.

- If the check box entitled *Confirm z values* item is selected, *WMS* will prompt for a z value every time a new vertex is created.

- If the *Interpolate for default z on interior* item is checked and a new vertex is entered in the interior of a TIN, the program will linearly interpolate a default z-value from the plane equation defined by the triangle containing the point.

- If the *Extrapolate for default z on exterior* item is checked and a new vertex is entered outside the TIN boundary, the program will extrapolate a default z-value by using a gradient based inverse distance weighted interpolation.

- The *X-Y tolerance* edit box shows the tolerance used for such TIN operations as removing duplicate vertices, insertion of breaklines, and dividing drainage boundaries. You should not need to change this value, but occasionally it becomes the only way to work around sticky numerical problems.

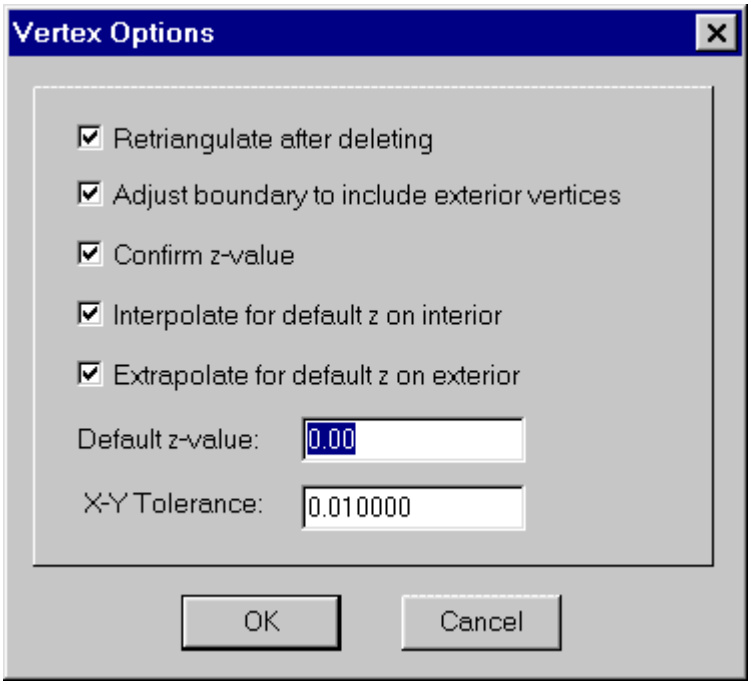


Figure 6-2 Vertex Options Dialog.

6.5.5 Lock / Unlock Vertices

Since it is possible to accidentally drag points, selected vertices can be "locked" to prevent them from being dragged or edited (using the *Edit Window*) by selecting the *Lock Vertices* command from the *TINs* menu. Any number of combinations of vertices can be locked or unlocked.

Locking and unlocking vertices provides a differentiation between points that are hard (measured data) and points that may be soft (interpolated or estimated data).

Selected vertices can be unlocked by selecting the *Unlock Vertices* command from the *TINs* menu. The status of each vertex, locked or unlocked, is preserved in the TIN file when TINs are saved to disk. Display options can be changed so that a distinction between ulocked and locked vertices is easily visible.

6.5.6 Scaling Vertices

The Scale Vertices command can be used to scale the x, y or z values of the TIN vertices. This can be used to convert from one set of units to another. However if your units are already in either feet or meters, then units of computed geometric parameters can be separated when issuing the *Compute Basin Data* command (See section 7.7.10).

6.5.7 Remove Duplicates

The triangulation algorithm assumes that each of the vertices being triangulated are unique in the xy plane, i.e. no two points have the same xy location. When a new set of points is imported to *WMS*, duplicate points should be removed by selecting *Remove Duplicates* from the *TINs* menu. Otherwise, *WMS* may abort when the points are triangulated. The tolerance for duplicate vertices can be set in the *Vertex Options…* dialog discussed previously.

6.6 Triangulation

A TIN can be constructed by triangulating a set of vertices. *WMS* connects the vertices with a series of edges to form a network of triangles. The resulting triangulation satisfies the Delauney criterion The Delauney criterion ensures that no vertex lies within the interior of any of the circumcircles of the triangles in the network (Figure 6-3).

As the triangulation process proceeds, adjacent triangles are compared to see if they satisfy the Delauney criterion. If necessary, the adjacent edge of the two triangles is swapped (the diagonal of the quadrilateral defined by the two triangles is changed to the other two vertices) in order to satisfy the Delauney criterion. This edge swapping process forms the basis of the triangulation algorithm.

When a new point is inserted into a TIN, the point is incorporated into the TIN and the edges of the triangles adjacent to the new point are swapped as necessary in order to satisfy the Delauney criterion. If the Delauney criterion is satisfied everywhere on the TIN, the minimum interior angle of all of the triangles is maximized. The result is that long thin triangles are avoided as much as possible.

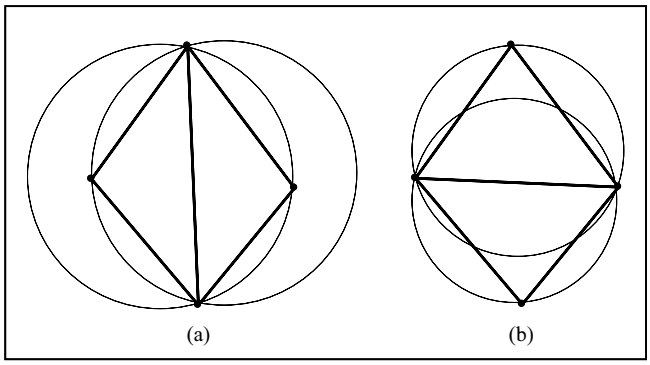


Figure 6-3 Two Adjacent Triangles Which (a) Violate and (b) Honor the Delauney Criterion.

It is important to note that the triangulation described above is used as a preliminary step to creating a TIN conditioned for basin delineation and is not sufficient in most cases for actually doing the drainage delineation (see section 6.6.2). Even if you begin with TIN data you will want to create another TIN using feature objects (see page 3-33).

6.6.1 Triangulation Options

Three methods or algorithms are used to triangulate points:

- The *convex hull* method computes the convex hull of the data points, triangulates the convex hull, and then inserts the interior points (non-hull points).

- The *enclosing triangle* method makes one large triangle that encloses the points, inserts the points into the triangulation, and then deletes all triangles connected to the three extra points used to make the enclosing triangle.

- The *enclosing triangle plus fill* method is the same as the enclosing triangle method except that after the extra triangles are deleted the concave regions on the boundary are filled in so that the boundary corresponds to the convex hull.

The *enclosing triangle* methods are faster than the *convex hull* method, but the *convex hull* method is the most robust. The *enclosing triangle* methods may not triangulate all of the points if the points are grouped in a long thin line or row.

The triangulation method can be specified by selecting *Triangulation Options* from the *TINs* menu. This dialog also allows the display of the triangulation process to be turned off and on. It is often informative to display the triangulation process as it proceeds. However there is a significant increase in the overall time to complete the triangulation when the display is on.

6.6.2 Triangulate

Vertices can be triangulated using the currently selected triangulation algorithm by selecting the *Triangulate* command from the *TINs* menu. It is important to recognize that the Delauney triangulation is not necessarily the best for performing drainage delineation because it does not insure that important linear features such as streams and ridges will be honored in the TIN as triangle edges. For this reason you should always use a TIN triangulated in this fashion as a "background" elevation source for creating a new TIN from a "conceptual" model of feature objects as described in section 3.2.8.

6.6.3 Subdivide TIN

The density of a TIN can be increased using the *Uniformly Subdivide TIN* command in the *TIN* menu. You will be prompted for a subdivision factor. This factor is then used to uniformly subdivide the TIN into subtriangles (Figure 6-4).

The *Uniformly Subdivide TIN* command can be used to "smooth" a TIN. When using a TIN for contouring, the contours are computed using a linear interpolation of the triangles. If the vertices are sparse, the contours are not smooth and the plot may not appear natural. The contours can be smoothed by copying the vertices to a scatter point set, subdividing the TIN into subtriangles, and interpolating the z values (or other data sets) from the scatter point set to the new vertices defining the subtriangles.

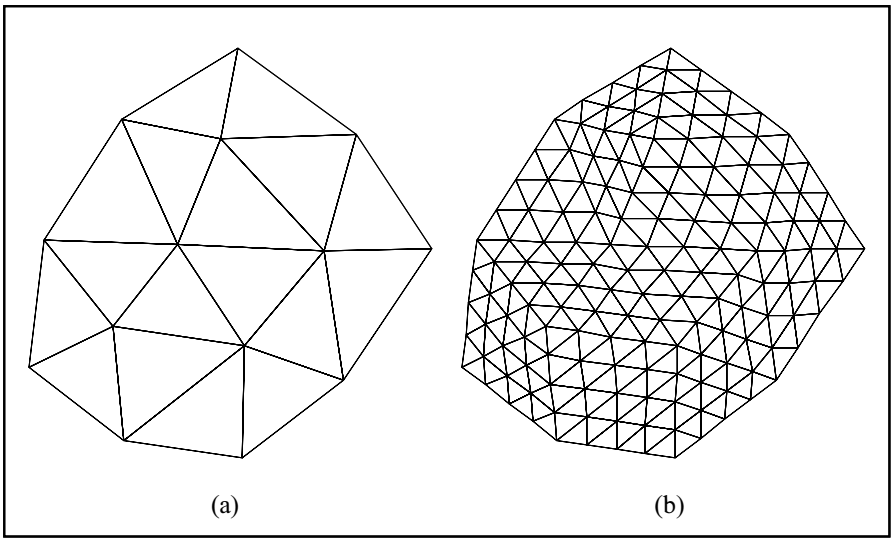


Figure 6-4 TIN (a) Before and (b) After Uniform Subdivision.

Subdivision and smoothing can be accomplished using the following steps:

1. Convert the TIN to a scatter point set using the *TIN -> Scatter Points* command in the *TINs* menu.

2. Subdivide the TIN by selecting the *Uniformly Subdivide TIN* command from the *TINs* menu.

3. Switch to the *2D Scatter Point* module and select an interpolation method using the *Interpolation Options* command in the *Interpolation* menu.

4. Select the *to TIN* command from the *Interpolation* menu. This creates a new elevation set for the TIN.

6.6.4 Merging TINs

You can merge a selected TIN with another TIN using the *Merge to Active* command in the *TINs* menu. This is particularly useful if you wish to merge a TIN generated from one program with a TIN derived from a background elevation source such as a DEM.

For example you may have surveyed data with a lot of detail for a part of your project (such as a roadway profile) and wish to combine that with a TIN derived from a DEM of the surrounding area. To do this you would first make sure that the TIN representing the background or surrounding area is the active TIN. Next you would select the TIN derived from the detailed survey or other data. You then choose the *Merge to Active* command. Figure 6-5 illustrates this process.

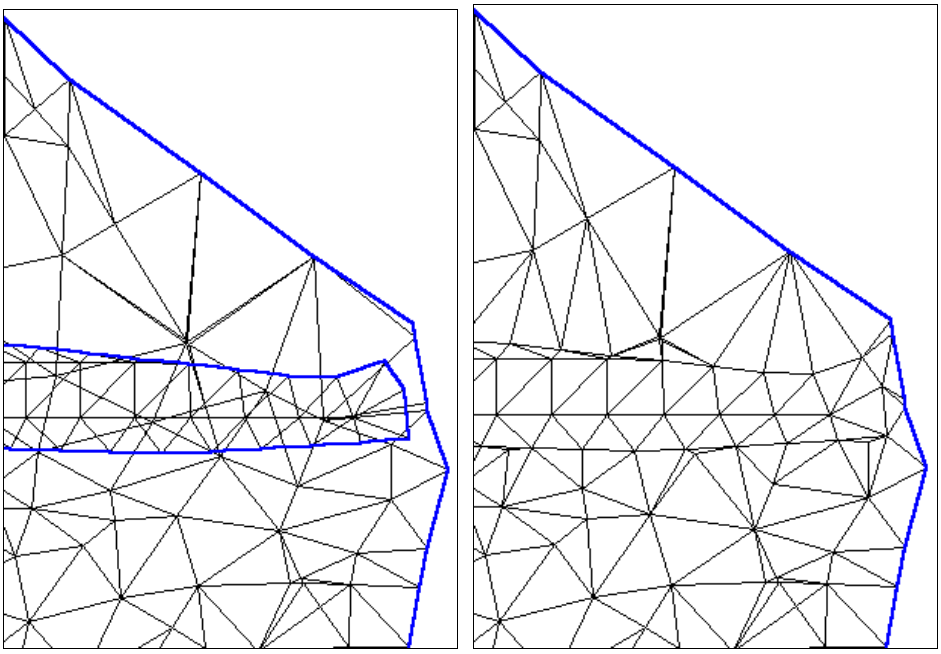


Figure 6-5 TIN Merge Results.

The following rules are adhered to when merging a selected TIN to the active TIN:

- The selected TIN is always merged into the active TIN and "overwrites" any overlapping data. In other words triangles and vertices in the active TIN that are overlapped by the selected TIN are deleted.
- All triangle edges in the selected TIN are preserved when merged into the active TIN.
- If the selected TIN overlaps any stream vertices on the active TIN the stream will be split. The stream will be cutoff wherever it "enters" the TIN selected for merging and begin again upstream where it "exits."
- Drainage data in the selected TIN that overlaps the active TIN will be lost.

6.7 Breaklines

A breakline is a feature line or polyline representing a stream channel, ridge or some other feature that you wish to preserve in a TIN. In other words, a breakline is a series of edges that the triangles should conform to as shown in Figure 6-6. Breaklines can be very useful when trying to eliminate unwanted pits on the interior of a TIN.

Breaklines can be processed using the *Add Breaklines* command from the *TINs* menu. Before selecting the command, one or more sequences of vertices defining the breakline(s) should be selected using the *Select Vertex Strings* tool in the *Tool Palette*.

6.7.1 Breakline Options

The *Breakline Options* option in the *TINs* menu brings up a dialog which allows you to specify either *Add points and preserve Delauney triangulation* options, or the *Swap edges* option for processing breaklines. When the *Add points and preserve Delauney triangulation* option is selected, new vertices are added to the TIN at necessary locations to ensure that the edges of the triangles will conform to the breakline. The elevations of the new vertices are based on a linear interpolation of the breakline segments. The locations of the new vertices are determined in such a way that the Delauney criterion is satisfied (see section above on triangulation). When the *Swap edges* option is selected, no new vertices are added to the TIN. The edges that intersect the breakline are swapped with the breakline to form triangles. The Delauney criterion is not checked or maintained.

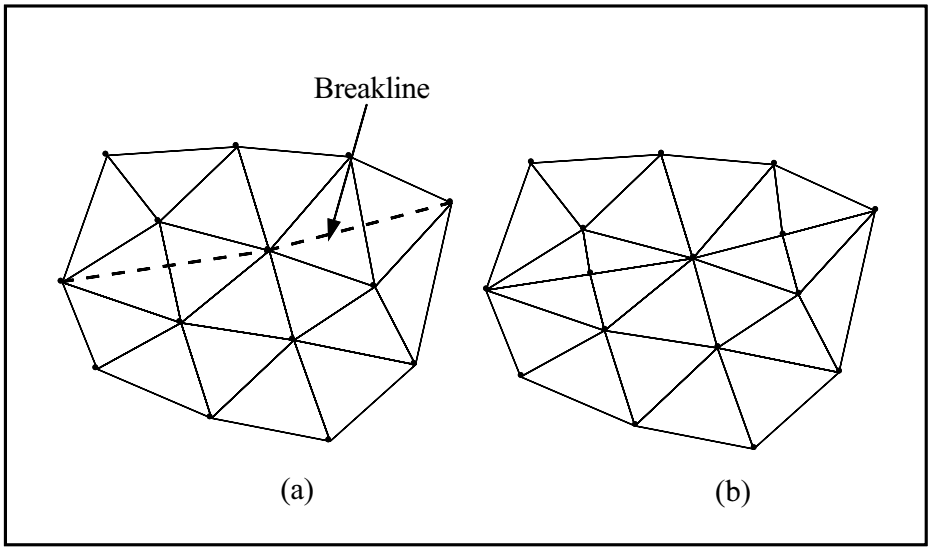


Figure 6-6 (a) TIN and Breakline. (b) TIN After the Breakline has been Processed.

6.8 Boundary Triangles

The perimeter of the TIN resulting from the triangulation process corresponds to or approximates the convex hull of the data points. This may result in some long thin triangles or "slivers" on the perimeter of the triangulated region. There are several ways to deal with the long thin triangles.

6.8.1 Selecting Boundary Triangles

The thin triangles can be selected and deleted using the normal selection procedures. There is also an option for selecting thin triangles when the *Select Triangles* tool is selected. If the *CTRL* key is held down, it is possible to drag out a line with the mouse. All triangles intersecting the line will be selected.

Another technique can be used to select long thin triangles on the perimeter of the TIN. By selecting the *Select Boundary Triangles* item from the *TINs* menu, the thin triangles on the perimeter of the TIN are automatically selected.

The *Select Boundary Triangles* command checks triangles on the outer boundary first. If the length ratio of the triangle is less than the critical length ratio, the triangle is selected and the triangles adjacent to the triangle are then checked. The process continues inward until none of the adjacent triangles violate the minimum length ratio.

6.8.2 Length Ratio

The critical length ratio for selecting thin triangles can be set by selecting *Length Ratio* from the *TINs* menu. The length ratio is defined as the longest side of the triangle divided by the sum of the two shorter sides.

6.9 Automated TIN Editing

After the initial triangulation (meaning after a TIN has been created from a conceptual model using feature objects and a background elevation source), it is possible that flat triangles, flat edges, pits, and other TIN anomalies may exist. Flat objects must always be removed before performing any type of drainage analysis.

Most of these problems occur because of a lack of resolution in the elevation data (it would be impractical though in most cases to increase the resolution to the point where these anomalies are eliminated). Most of the problems can be fixed by a combination of inserting additional vertices, deleting some vertices, smoothing elevations, adding breaklines, or swapping edges.

In order to automate the editing process as much as possible, several different techniques are provided to remove flat objects and better infer stream networks.

- Interpolating Flat Triangles/Edges - Inserts new points in flat triangles and adjusts the elevation of the new points by using an interpolation technique. This method works well when there is a small number of clustered (2-10) flat triangles. However, when large regions of flat triangles exist, the TIN filtering should be used before trying to remove flat objects.
- Smoothing Flat Triangles - This method works well if you have isolated flat triangles that are not clustered (no more than 2 or three adjacent flat triangles). With this method flow paths are computed into and out of the flat triangle to infer the general direction of flow. Edges are then swapped to maintain the inferred flow regime.
- Smoothing Pits - Pits may be the result of isolated depressions within a terrain surface. They can be removed by swapping neighboring edges or slightly adjusting vertex elevations to simulate "filling" of the depression.
- TIN Filtering - Typically large flat regions occur because of rounding errors in the calculation of the elevations, and TIN filtering uses an averaging technique to appropriately adjust elevations relative to surrounding vertices. If there are too many flat triangles grouped together (such as is the case when a reservoir exists that has a constant surface elevation), then you may need to delete several of the interior vertices before doing the filtering.

The details and options available for each of these methods are discussed in the following sections.

6.9.1 Interpolating Flat Triangles

When the *Interpolate Flat Triangles* command in the TIN menu is issued, *WMS* computes the differences between the elevations of the flat triangles and the elevations of the surface defined by the IDW quadratic interpolation scheme. This technique is used because of its ability to accurately infer important terrain features such as pits, peaks, streams, and ridges. The difference between the flat surface and the interpolated surface is referred to as the "deviation" of the flat triangles. The deviation is computed at regularly spaced points on the interior of each flat triangle. The subdivision factor in the *Interpolation Scheme* dialog controls the level of subdivision or the number of interior points. The point in each flat triangle with the maximum deviation is assumed to represent the maximum for that triangle.

Once the deviations are determined, *WMS* locates the flat triangle whose deviation is the maximum. A new point is added at the XY location of the maximum deviation. The elevation of the new point is computed using the IDW quadratic interpolation scheme. The new point is inserted into the TIN, and the TIN is adjusted locally to accommodate the new point. Many times the insertion of a new point in a flat triangle combined with the local retriangulation of the TIN results in the removal of several flat triangles. The list of flat triangles is updated, the flat triangle with the next largest deviation is found, and the process is repeated. By inserting new points in this fashion, the minimum number of new points will be added in the best possible locations to infer local minima and maxima such as pits, peaks, streams, and ridges.

Once all flat triangles have been eliminated, further processing to remove flat edges and pits is done. Prior to performing the Remove Flat Objects command all TIN vertices are locked. Any new vertices created as part of this process are set to ulocked status. When completed you will be able to distinguish the new vertices from the original by observing the ones that are unlocked. You may wish to unlock all vertices for further editing.

6.9.2 Smoothing Triangles

If you have a small number of isolated flat triangles (no more than 2 or three adjacent/connected flat triangles), then the *Smooth Triangles* command is the ideal way to eliminate them. Each flat triangle fits one of the following classifications and is fixed as described.

1. If flow from all three edges is into the triangle then this triangle is a pit or depression region. This class of flat triangle is fixed by creating a vertex at the centroid whose elevation is 1/1000th lower than the

vertices of the flat triangle. You may need to use the Smooth Pits command described below to further eliminate the pit just created.

2. If flow from all three edges is out of the triangle then this triangle is a peak region. This class of flat triangle is fixed by creating a vertex at the centroid whose elevation is 1/1000th higher than the vertices of the flat triangle.

3. If flow from two edges is out of the triangle and flow from one edge is into the triangle then the fix is to lower the elevation of the vertex between the two edges flowing out by 1/1000.

4. If flow from two edges is into the triangle and flow from one edge is out then the fix is to swap the edge from which flow is away.

It should be noted that if one of the triangle edges is adjacent to a flat triangle flow from that edge is considered to be "into" the triangle. This may not actually be the case and is the reason why this method should not be used if there are large "clusters" of flat triangles.

6.9.3 Smoothing Pits

The *Smooth Pits* command adjusts the elevations of pits in order to remove them. For each pit the two next highest (elevations) of adjacent vertices are located and the elevation of the pit is set to the average of these two elevations. Flat triangles should be removed before using this command.

6.9.4 Filtering A TIN

Typical gridded data such as USGS quads contain elevations that have been rounded to the nearest foot or meter. This rounding process creates a stair-step pattern of elevations and often results in large areas of flat triangles. Removing large areas of flat triangles cannot be done with the *Remove Flat Objects* command because interpolated surface values in large flat areas to not deviate from the flat triangles themselves. Thus any new points added would only create more flat triangles. In such cases, filtering must be done prior to removing flat objects.

When a TIN is filtered, the elevation of each unlocked vertex is averaged with its neighbors. Locked vertices are used to compute the average value, but only unlocked vertices are actually modified. This averaging process blends local discontinuities in elevation, thus removing the original stair step pattern and smoothing the TIN. You can take advantage of the different ways locked and unlocked vertices are treated to influence how a TIN is filtered. For example, prior to filtering you can manually adjust the value of a vertex and then lock it. Filtered elevations will then be biased towards the elevation of the locked vertex.

Different filtering options, as described below, can be set to control how filtering proceeds. They can be set in different combinations until a desired result is reached. It is often useful to display contours and flat triangles to help guide the filtering process.

Filter TIN

The *Filter TIN* command filters the TIN using the current set of filtering options as described below.

Restore Elevations

The *Restore Elevations* command changes the elevations of the vertices back to the last stored values. The first time a TIN is filtered the elevations are stored by default. Thereafter, if you wish to "remember" a set, you must explicitly store them from inside the *Filter Options* dialog. Once the filtering process is complete (commands other than filtering are chosen), the status of the stored elevations is no longer valid, i.e., you cannot restore the elevations.

Filter Options

Filter options control how a TIN is modified when issuing the *Filter TIN* command. Different combinations of the settings produce somewhat different results.

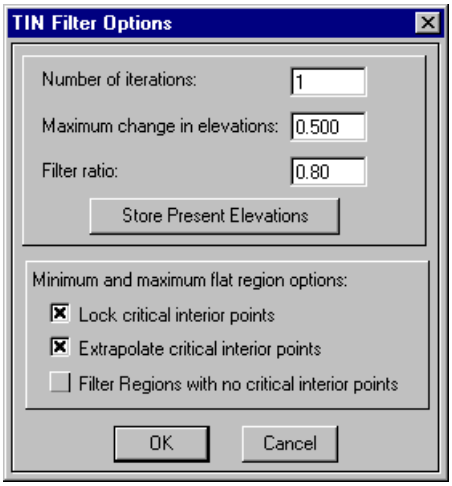


Figure 6-7 TIN Filter Options Dialog.

Number of Iterations

During the filtering process, the elevation of each unlocked vertex is redefined as the average of neighboring elevations. The number of iterations, or times a new elevation is computed for each vertex, can be set with this option. Because of the effects of the other options, filtering a TIN n times with an iteration of 1 does not necessarily produce the same result as filtering a TIN

once with an iteration of n. Since elevations can be reset, this number can be experimented with, but typically you will want to start

Maximum change in elevation

This value determines what the maximum change in elevation can be during the filtering process. For example, in the case of USGS data where elevations have been rounded to the nearest meter, an appropriate maximum change value would be 1/2. This would guarantee that even if the averaged elevation is greater than 1/2 meter, the new elevation only changes by 1/2. This value can be set to an arbitrarily high value so that the average elevation everywhere results from filtering, without putting a limit on the amount an elevation can change.

Filter Ratio

The filter ratio "softens" the filtering effect by changing the elevation by a percentage of the difference in the current Z value of a vertex and the computed average during filtering.

Store present elevations

The set of elevations restored when using the *Restore Elevations* command depends on the last time elevations were stored. By default, the first time filtering is done, all elevations are stored. Thereafter, if you wish to "remember" a current set of elevations, you should use this command to store them. Once the filtering process is complete (commands other than filtering are chosen), the status of the stored elevations is no longer valid, i.e., you cannot restore the elevations.

Minimum and Maximum

One of the primary functions of filtering is to remove large flat regions on a TIN. When a large flat region is located a check is made to see if the vertices bordering the flat region are all greater than or all less than the elevation of the flat region (i.e., a local minima (pit) or a local maximum (peak) flat region). In this case points which are furthest from the boundary (most likely to be the pit/peak points) can be locked, extrapolated by the maximum value, or both. Such points are referred to as critical interior points. By locking these points, their elevations will not be altered during the filtering process. Locking the points is important because the entire flat region would just move up or down without inferring the pit/peak. Extrapolating to the maximum value gives the pit/peak more definition.

Some small flat regions do not have any interior points, i.e., the only points in the flat region lie on the boundary. Such areas can either be processed or not depending on the status of the "filter regions with no critical interior points" toggle. As described above, filtering such regions may only serve to slightly

raise or lower the flat region. In such cases *Remove Flat Objects* will more accurately infer local minima and maxima and you may wish to skip over these regions during the filtering process.

6.9.5 Edge Swapping

TINs are generated in *WMS* using the Delauney criteria. This method creates a set of triangles which are as equiangular as possible, and while this generally creates a good terrain surface, it does not ensure that all important hydrologic features such as streams and ridges will be honored with triangle edges. A classic problem which occurs and inhibits drainage analysis is the "false dam". A false dam occurs during the triangulation process when an edge straddles a natural channel, forming a dam in the bottom of the channel as shown in Figure 6-8a. False dams are easily corrected by swapping the triangle edge ab to cd as illustrated in Figure 6-8b. Triangle edges are swapped using the *Swap Edge* tool and clicking on the edge which needs to be swapped.

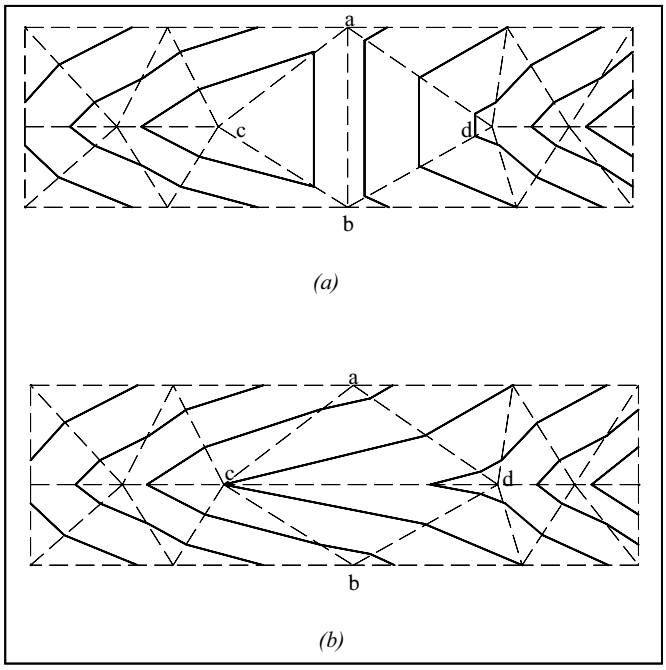


Figure 6-8 False Dam Correction.

6.10 TIN Boundary -> Polygon

The *TIN Boundary -> Polygon* command in the *TINs* menu can be used to create a boundary feature polygon from the vertices on the boundary of the

TIN. Such a polygon could then be used to create a grid from a feature polygon using the *Create Grid* command found in the *Feature Objects* menu.

6.11 TIN -> Scatter Points

The *TIN -> Scatter Points* command creates a 2D scatter point set from the TIN. One data point is created for each of the vertices in the TIN. A data set is made of each from the elevations of the TIN vertices. This command can be used as part of the process to create smooth contours by subdividing the TIN and then interpolating from the newly created scatter points.

6.12 TIN->DEM

You can convert a TIN to a DEM using the *TIN->DEM* command. When converting a TIN to a DEM *WMS* estimates default values for the xy coordinates of the lower left corner of the DEM, the DEM grid cell size, and the number of rows and columns. These parameters can be customized by editing the respective values in the *Convert TIN to DEM* dialog shown below. DEM grid cell elevations are computed by interpolation from the TIN vertices using either a linear interpolation or an inverse distance weighted method.

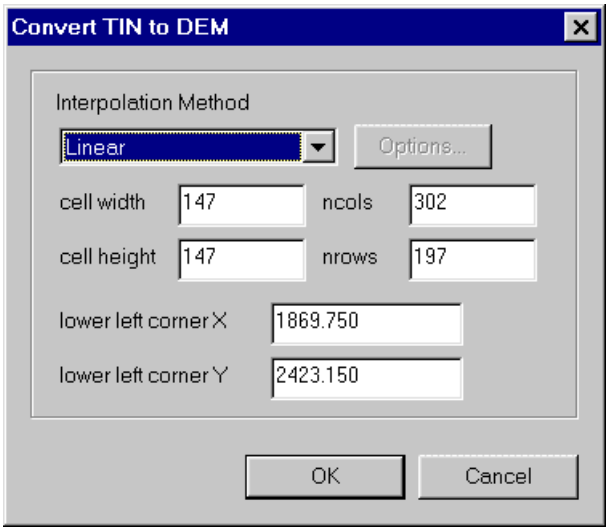


Figure 6-9 Convert TIN to DEM Dialog.

CHAPTER 7

Drainage - TINs

7.1 Introduction

A TIN terrain model can be used in *WMS* to delineate stream networks and drainage basin boundaries. Since the terrain model is an accurate geometric description of the watershed, parameters such as areas, slopes, and flow distances can automatically be computed. This terrain model then serves as a map to guide entry of all data necessary to run *HEC-1, TR-20,* or other hydrologic analysis programs.

Previous chapters described the necessary tools needed to prepare a TIN for drainage analysis. It should be emphasized that it is highly recommended you prepare your TIN for drainage analysis by using feature objects with a background elevation source. The elevation source could be TIN or DEM, but you should retriangulate using feature objects as guides in order to insure that stream edges are honored by triangle edges (see page 3-33). This chapter will deal with the specifics of defining the sub basins of a watershed.

The first process in performing drainage analysis is to edit the TIN where necessary. Flat triangles, flat channel edges, and flat ridge edges must all be eliminated before trying to delineate stream networks and basin boundaries. Automatic editing procedures, such as TIN filtering and removal of flat objects, should be used. In addition, manual insertion of breaklines, the addition of new points, and edge swapping can aid in removing anomalies which are introduced into the TIN as a byproduct of the triangulation process. With the TIN properly edited, stream networks and drainage basins can be defined as preparation for defining a complete hydrologic analysis.

7.2 Tool Palette

Since the drainage analysis options are found in the TINs module, the tools are the same as defined in the previous chapter.

7.3 Drainage Display Options

The display options dialog box (Figure 7-1) accessed from the *Drainage* menu allows control over the display of entities related to drainage analysis.

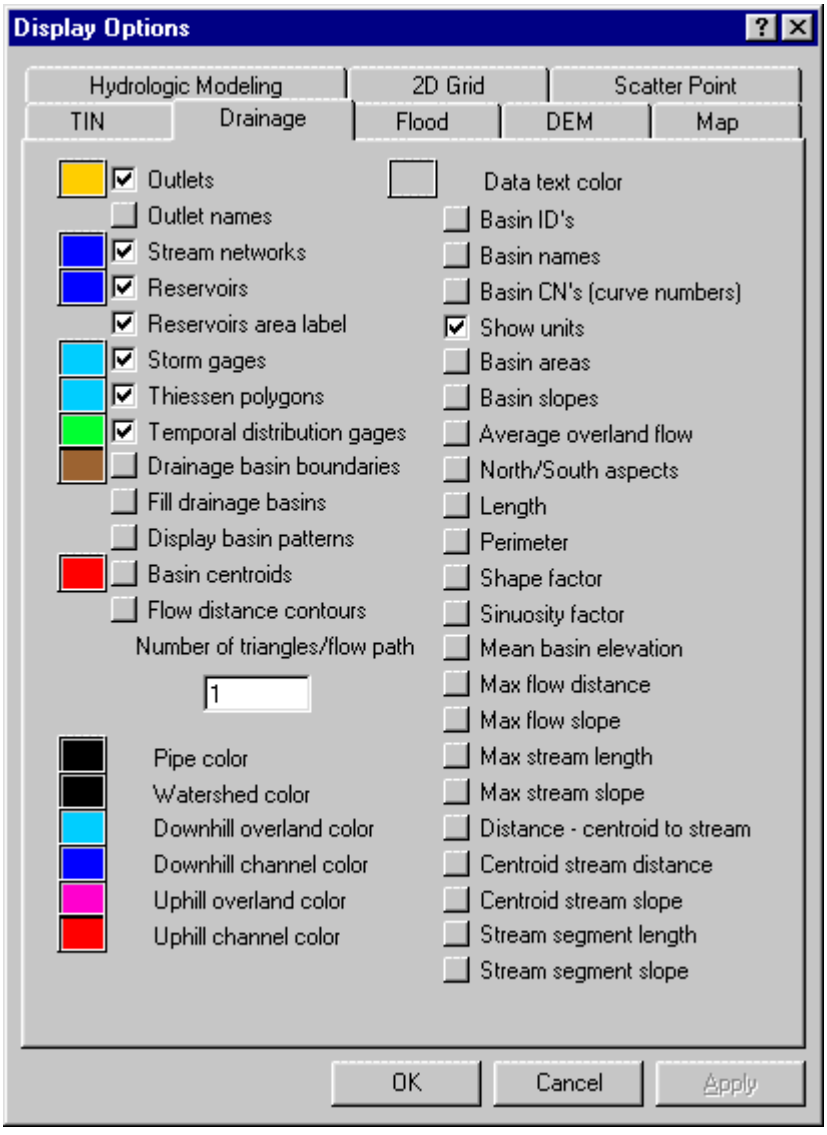


Figure 7-1 Drainage Display Options Dialog.

7.3.1 Outlets

If the *Outlets* option is set, all outlet vertices will be displayed. Outlet vertices correspond to points on the TIN where a watershed or sub basin outlet exists. When issuing the *Find Default Outlets* command from the *Streams* menu, all pits and points on the exterior of the TIN where channel edges exit are stored as outlets. After generating stream networks, additional outlets may be added at branches or other points in the stream such as gaging stations.

7.3.2 Outlet Names

The names assigned to the outlets can be displayed next to the outlet by toggling this option on. The default name for an outlet is the ID number. Names are only important when creating *HEC-1* analysis files. They are used to identify hydrographs which are combined or routed.

7.3.3 Stream Networks

If the *Stream Networks* option is set, all stream networks will be displayed. Stream networks are stored as consecutive channel edges and can be displayed each time the display is refreshed.

7.3.4 Pipe Color

When a stream segment is defined between two vertices that belong to the same triangle edge it is considered to be an open channel. However, underground channels, or pipes, can be defined by creating a stream between two vertices which belong to separate triangles. In such cases overland flow passes over the “stream” segment since it acts as a pipe. The *Pipe color* can be set to display segments of this type.

7.3.5 Reservoirs

The *Reservoirs* options can be used to display triangles defined as reservoirs. If basin data has been computed then the area labels for each reservoir can be displayed.

7.3.6 Gages

When gages are defined for *HEC-1* models, icons displaying the location of the gage and a Thiessen network defined by the gages can be displayed by setting the *Storm gages* options. These gages can also be created using the Rain gage coverage type. Temporal distribution gages are used to define gages that contain temporal information about a storm as well as rainfall

depths. They are not triangulated as part of the Thiessen network unless they are also defined as storm total gages.

7.3.7 Thiessen Polygons

When a set of gages have been defined this toggle can be turned on to show the accompanying thiessen polygon network.

7.3.8 Basin Centroids

If the *Basin Centroids* option is set, all basin centroids will be displayed. The basin centroid is also used as the initial location for displaying labels and hydrograph icons.

7.3.9 Drainage Basin Boundaries

If the *Basin Boundaries* option is set, all basin boundaries will be displayed. Basin boundaries are computed using the *Define Basins* command from the Drainage menu.

7.3.10 Fill Drainage Basins

If the *Fill Drainage Basins* option is set, all basins will be displayed using a different color and/or pattern. This provides a better graphical image than simply displaying boundaries. If the Display Basin patterns options is selected then a hathching pattern is displayed to distinguish adjacent basins that share the same color.

7.3.11 Basin IDs

The basin ID may be displayed by toggling this option on. Basin IDs are used to identify a basin with its associated values when exporting drainage data.

7.3.12 Basin Names

The name assigned to the drainage basin can be displayed at the basin centroid by toggling this option on. The default name for a basin is the ID number. Names are only important when creating *HEC-1* analysis files. They are used to identify the hydrographs of each basin.

7.3.13 Basin CN's

The curve number for each basin can be displayed by toggling this option on. Drain data must have been computed for CN labels to be placed at the basin centroids.

7.3.14 Number of Triangles/Flowpath

This option effects how the *Draw Flow Patterns* command works. By default, a flow path for each triangle is generated when this command is run. However, for large TINs it is not necessary, and very time consuming to draw a flow path for each triangle. Therefore, the number of triangles which have there flowpaths drawn can be reduced by setting this value higher.

7.3.15 Watershed Color

This color box allows the specification of color used when performing the *Draw Watershed* command found in the Drainage menu.

7.3.16 Downhill Overland Color

This color box allows the specification of color used when drawing flow paths which are downhill and overland (paths of maximum downward gradient).

7.3.17 Downhill Channel Color

This color box allows the specification of color used when drawing flow paths which represent channel flow.

7.3.18 Uphill Overland Color

This color box allows the specification of color used when drawing flow paths which are uphill and overland (paths of maximum upward gradient).

7.3.19 Uphill Channel Color

This color box allows the specification of color used when drawing flow paths which represent "ridge" flow.

7.3.20 Data Text Color

This color box allows the specification of text color, size, and font used when displaying basin attributes. No noticeable effect occurs until these basin labels are actually displayed. It is especially useful after changing the color of the background.

7.3.21 Geometric Attributes

A text string for any of the computed geometric attributes can be displayed by toggling the option on.

7.4 Outlets

An outlet by definition in *WMS* is a point that defines a confluence. It is the point where a sub basin ends and a routing reach begins. Default outlets are local minima or pits on the interior and stream exit points on the exterior of the TIN. Whenever an outlet point is added, a drainage basin for that outlet is also created. If the outlet corresponds to a stream branching point, then a drainage basin for each stream branch is created. For this reason the default outlets are not always sufficient. Outlet points can be added and deleted in order to define the sub basins of a watershed. All outlet commands are found under the streams menu.

7.4.1 Find Default Outlets

This command automatically finds all pits (local minima) on the interior of the TIN and channel edge exit points on the exterior of the TIN and adds them as outlets. These outlet points can then be used to define drainage basins directly or to create streams. When creating streams, a separate network is initiated at each outlet.

7.4.2 Add Outlets

The *Add Outlets* command will add all selected vertices as outlet points. A set of vertices must be selected for this command to work.

7.4.3 Delete Outlets

The *Delete Outlets* command will remove selected outlet points. Outlet points are selected using the *Select Vertices* tool. If a vertex that is not an outlet point is in the selection list, then no action is taken for that vertex. When an outlet is deleted, the area or triangles associated with that outlet's drainage basins are reassigned to the next downstream basin.

7.4.4 Select Branching Nodes

It is often desirable to add outlet points at a large number of stream branches. Therefore, this command allows such vertices to be automatically selected. This command only selects the vertices and does not actually create outlets from them. In order to create outlets from these vertices, the *Add Outlets* command should be performed following selection. If some of the branching

vertices should not be added as outlets, they can be deselected by clicking on the vertex while holding the *SHIFT* key down. If there are a large number of stream branches and only a small number are to be added, they can be selected individually using multi-selection techniques.

7.5 Streams

Streams can be defined for a TIN by linking together consecutive channel edges. Streams can be defined by manually connecting the edges.

Automatically delineating a stream network can only occur if all channel edges have been explicitly incorporated into the TIN. This can be done using the TIN editing techniques described in the previous chapter along with breaklines, swapping edges, and point insertion. You can verify that a continuous set of channel edges exist by toggling on the display of channel edges. Stream networks are automatically delineated for each outlet point in the TIN.

Because TINs are created from a sparse set of points, it is often difficult to explicitly define channel edges, particularly in urban areas where well defined channels may not even exist. For this reason, a stream can also be created by manually selecting a set of vertices which defines a channel. This method also allows a street or storm drain to be incorporated into the TIN and used as part of the "stream network".

7.5.1 Create Streams

When the *Create Streams* command is chosen, a stream network for the current selected string is created. The vertex string is made by connecting points which are known to lie in a stream, street, or other drainage structure. If the entered vertex string crosses triangle edges, the user will be asked if he wishes to insert a breakline. The breakline is inserted by creating new points where the line crosses a triangle edge. The elevation of the new points is determined by linear interpolation along the edge. Inserting the breakline in this fashion alters the topology without affecting the geometry.

Since flow through a stream network is defined strictly by the “linked” set of vertices, a downstream vertex does not even have to be lower than the upstream vertex. However, care should be taken to ensure that the general direction of the stream is downhill. The intent of creating streams in this fashion is to eliminate the need of defining a continuous set of channel edges with the TIN editing techniques described above. This type of stream creation is particularly important when doing basin delineation for urban areas.

7.5.2 Create Pipe

When the *Create Pipe* command is chosen, a pipe connecting stream nodes in the current selected string is created. A pipe can be used to create a "stream" from any stream node to another without enforcing a continuous set of triangle edges between the two stream nodes. Pipes differ from streams in that when a flow path intersects a stream it then continues down the stream from node to node, whereas when a flow path intersects a pipe it continues overland across the pipe.

7.5.3 Delete Stream Segments

Deletes the segment of the stream between two selected stream nodes. An outlet point is inserted at the upstream node. If a single stream node is selected this command deletes the portion of the stream network from the selected stream node upward, including the selected stream node.

7.5.4 Delete All Streams

The *Delete All Streams* command deletes all stream networks. Outlet points will remain after the streams are deleted.

7.5.5 Display Stream Profile

The Display Stream Profile command is used to display elevation profile between two selected stream nodes. Because of the limited resolution of most elevation data sets it is difficult to get a continuous set of stream bed elevations. Within this dialog you can select and edit the elevation of individual stream nodes or you can select two different stream nodes from the profile plot and linearly interpolate the elevation of all nodes in between.

When using this command you must select at least two stream nodes (see section 2.9.2 for multi-selection of TIN vertices) and the second one selected must be upstream of the first.

7.6 Reservoirs

A set of triangles can be grouped together to create a reservoir. When creating reservoirs an outlet point must be specified for the triangles so that any flow path intersecting a triangle belonging to a reservoir can be routed directly to the outlet. The *Add Outlets* command is used to convert a stream node into an outlet point.

7.6.1 Reservoirs and Storage Capacity Curves

The *Create Reservoir* command in the *Streams* menu is used to create a new reservoir and/or a storage capacity curve for a selected outlet point. The dialog shown in Figure 7-2 allows you set up different options for generating reservoirs and storage capacity curves.

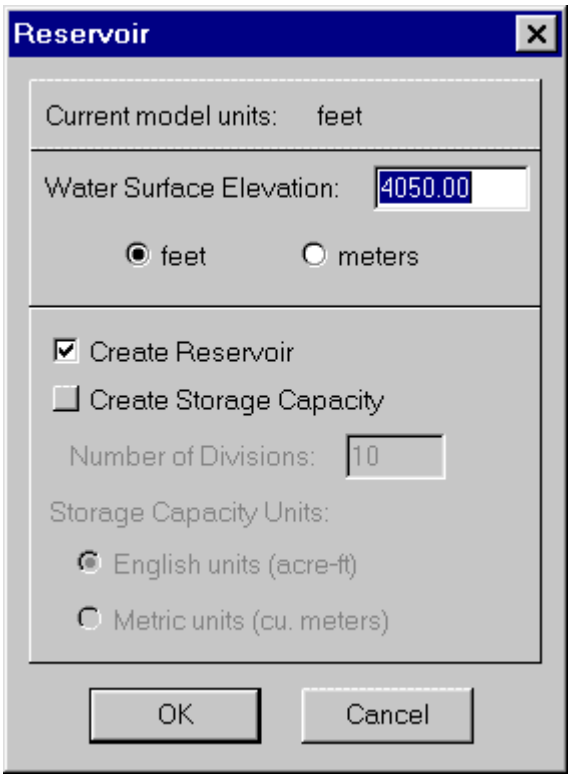


Figure 7-2 Reservoir Dialog

Creating a Reservoir on a TIN

If the *Create Reservoir* option is specified then all triangles which belong to the same basin as the selected outlet point and have an elevation less than or equal to the specified water surface elevation are converted to reservoir triangles. Reservoir triangles still belong in the same drainage basin, but when a flow path intersects one of these triangles it is directed to the reservoir outlet. Triangles with vertices above and below the water surface elevation are split along the waters surface elevation contour of that triangle.

The water surface elevation being used must be entered in the correct model units. The current model units setting is shown just beneath the edit field.

Creating a Storage Capacity Curve

If the *Create storage capacity curve* option is chosen then a relationship between elevation/area/volume is computed using the TIN geometry. These

three curves are stored in the storage list used by the time series editor so that they can be used later to define routing in one of the supported hydrologic models or in the detention basin calculator (section 15.6). *WMS* computes these relationships by beginning at the outlet elevation and incrementing the elevation by the number of specified divisions until the specified water surface elevation is obtained. At each increment the area between that elevation and the outlet elevation is computed and then volumes between adjacent surface elevations are computed using the conic method (see section 15.6). The storage capacity (elevation, area, volume) data can be stored in either English or metric units.

7.6.2 Deleting Reservoirs

An entire reservoir can be deleted by selecting the reservoir outlet using the *Select Vertices* tool and issuing the *Delete Selected Reservoir* command.

7.7 Drainage

Drainage basins are defined by starting a flow path from the centroid of each triangle and then classifying it according to the outlet first encountered by the flowpath. Once the basins are defined, many attributes associated with the basin such as area, slope, and maximum flow distance can be determined, displayed, or written to a file.

7.7.1 Define Basins

The *Define Basins* command assigns each triangle in the TIN to a drainage basin. This is accomplished by initiating a flow path from the centroid of each triangle and "flowing" down until an outlet point is encountered. The triangle is then assigned the appropriate basin ID. The boundaries may appear rough or jagged because each triangle is assigned according to the flow from its centroid, when in fact the triangle may actually straddle basin boundaries. Boundaries may be corrected by issuing the *Refine Boundaries* command. The drainage basin boundaries option in the *Drainage Display Options* dialog is automatically set when defining basins.

7.7.2 Refine Boundaries

After the initial definition of drainage basins, many of the boundaries are rough or irregular. Triangles straddling true basin boundaries can be split using the *Refine Boundaries* command. This process is accomplished by tracing paths of maximum upward gradient along boundaries, splitting triangles when the path crosses over them, and then reassigning all affected triangles to their new basins. The process is displayed graphically.

7.7.3 Correct Split Flow Vertices

Split flow vertices can usually be corrected by finding a channel edge leading into the split flow vertex, and swapping it. This edge swapping can be done automatically using the *Correct Split Flow* command from the *Drainage* menu. If the edge cannot be swapped without creating overlapping triangles the split flow vertex will not be eliminated and you will have to add new vertices, adjust elevations, swap multiple edges or some other form of manual editing technique.

7.7.4 Merge Basins

Selected basins can be merged together using the *Merge Basins* command. In order to select drainage basins, the *Select Drainage Basins* tool must be active. For each upstream branch of an outlet point, a drainage basin is automatically created. This command allows you to combine the basins for a given outlet. In order to merge basins, they must be adjacent to each other and belong to the same outlet.

7.7.5 Split Basins

The *Split Basins* command subdivides selected basins into separate sub basins, if possible. By default, a separate drainage basin for each upstream branch of an outlet point is created. If the drainage basins of a given outlet have been merged together, they can be split again using this command.

7.7.6 Delete All Basin Data

This command is used to erase from memory all of the currently defined outlet points, stream networks, and drainage basins.

7.7.7 Delete Null Basin Triangles

The *Delete Null Basin Triangles* command can be used to delete all triangles whose flow path does not encounter an outlet. Before defining drainage basins all triangles are classified as belonging to the null drainage basin. After defining drainage basins some triangles still belong to this null basin since they do not contribute flow through any of the given outlets. They should not be deleted if further editing is to be done. However once all sub basins have been properly defined, they can be deleted in order to reduce the size of the model to the region of interest.

7.7.8 Draw Flow Patterns

This command is used to draw a flow path for each triangle in the TIN. While it does not store stream networks and basin boundaries in memory, it aids in the initial understanding of the terrain model and helps identify regions which need editing before the actual creation of outlets, stream networks, and drainage basins takes place. For large TINs, the number of triangles which have their flow paths drawn can be reduced by changing the number of triangle/flowpath value in the *Drainage* tab of the *Display Options* dialog. Overland and channel flow are represented by the downhill overland color and downhill channel color as specified in the *Drainage* tab of the *Display Options* dialog. If a basin is selected prior to issuing the command then flow paths will only be drawn for the triangles that are part of the selected basin.

7.7.9 Draw Watershed

The entire watershed for an outlet point (i.e. the sum of all upstream sub basins) can be displayed by first selecting the outlet point and then choosing the *Draw Watershed* command. The *Select Vertex* tool must be active in order to select the desired outlets. If basin data has been computed the area enclosed by the watershed boundary will be printed in the help window.

7.7.10 Compute Basin Data

After defining basin boundaries, attributes such as basin areas and slope and stream lengths and slopes can be computed using the *Compute Basin Data* command. These are all geometric parameters used in defining basins and routing networks in *HEC-1* and *TR-20*. If the TIN is edited or sub basin configuration changed, the drainage data must be recomputed using this command.

Once computed, they can be displayed along with the basins in the *Graphics Window* or written to a file using the *Export File* command found in the *File* menu. By default, basin ID's and areas are displayed after computing drainage data. The other attributes can be toggled on for display in the *Basin Attributes* dialog accessed from within the *Drainage Display Options* dialog.

Units

When the *Compute Basin Data* command is given the *Model and Parameters Units* dialog (Figure 7-3) appears allowing you to specify the current units of the model (TIN vertices) and the units you wish to use for computed parameters. TIN vertices must either be in feet or meters, but the computed areas and lengths can be given separate units. If you wish to change the units of computed parameters at a later point you must recompute the basin parameters with different selections. Using the appropriate conversion factor,

the *Scale Vertices* command (see section 6.5.6) can be used to convert the TIN vertices from one set of units to another.

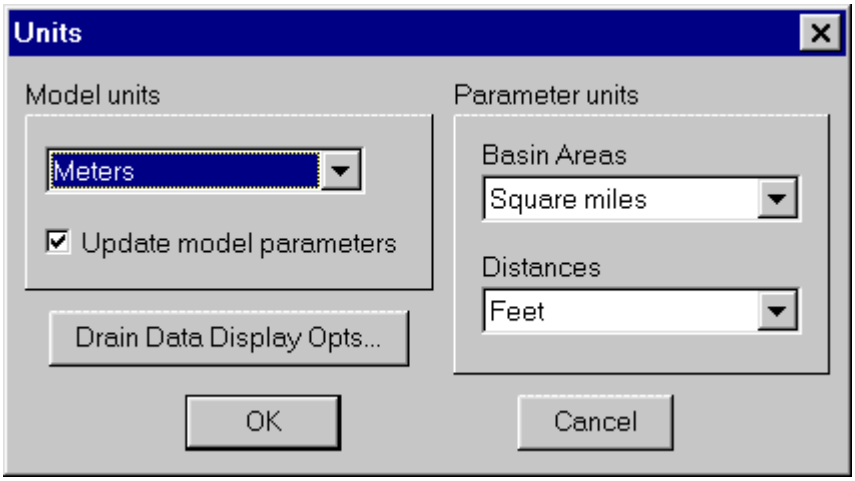


Figure 7-3 Parameters Units Dialog

7.7.11 Computing Area Between Elevations

The *Compute Area Between Elevations* command is useful for determining areas in different elevation zones as part of a snow melt analysis. This operation can also be done when defining snow melt parameters for *HEC-1* (see section 10.10.1). Model units are assumed to be either in feet or meters and subsequent areas are converted to square miles or square kilometers according to the metric flag set in the *HEC-1 Job Control* dialog.

This same procedure is also useful for determining storage capacity curves. It is incorporated into the procedure outlined in section 7.6.1, but can also be used to manually determine an area-elevation curve.

7.8 Converting Drainage Data to Feature Objects (GIS Data)

In order to provide a way to export watersheds delineated from TIN data sources to GIS they must first be converted to feature objects. This can be done using the *Drainage Data -> Feature Objects* command. When performing this operation the TIN is deleted and the drainage boundary and stream network are converted to feature polygons and feature arcs. The Export command from the File menu can then be used to save the feature objects to a shapefile so that the information can easily be transferred to a GIS (see section 2.8.7). All of the drainage data computed/entered for the TIN basins and junctions (areas, lag times, slopes, and other hydrologic modeling parameters) are automatically transferred during the conversion process.

CHAPTER 8

Flood Plain Analysis

8.1 Introduction

In addition to stream network and drainage basin delineation, *WMS* can also be used to perform flood plain delineation. Stage values (water depth) are defined by the user at selected locations in the TIN. A smooth surface representing the water surface is then constructed which interpolates these stage values. This surface is then intersected with the triangles in the TIN, and the resulting set of edges defines the flood plain.

The water surface is constructed from the stage values using a special type of inverse distance weighted (IDW) interpolation. For each vertex where a stage value has been assigned, a plane is constructed which interpolates the water surface elevation at the vertex. The slope of the plane approximates the water surface elevation at the vertex. These planes are called gradient planes, and the surface is constructed by "blending" the planes together to form a continuous smooth surface which honors the stage values. At each point of the surface, the elevation is defined by averaging the elevation of the gradient planes at that point. During the averaging process, more weight is given to planes whose vertices are nearest the interpolation point.

An important step in the surface construction process is the computation of the slopes or gradients which define the planes. The gradient should approximate the local trend of the water surface. If stage values are assigned to nodes of a stream network, the network is used to approximate the gradients. A plane is defined for each segment of the stream. The plane passes through the stage values on each end of the stream, and the normal to the plane lies in a vertical plane passing through the stream edge. This ensures that lines lying in the

plane which project outwards perpendicular to the plane are horizontal. The normals of the planes associated with each segment adjacent to a stream node are averaged to produce the normal, which defines the gradient plane at the node. If a stage value is assigned to a vertex which is not part of a stream, a plane with zero slope is assigned to the vertex.

The intersection of the water surface and the TIN is computed in such a manner that the water surface is not projected through a continuous ridge and into a neighboring valley, unless there is a spill point or passageway into the neighboring valley. This is accomplished by starting the search for the intersection at the triangles adjacent to the nodes where a stage value is defined and progressing outward. The propagation ceases when the intersection is found.

Stage values may be entered interactively and edited when necessary. Stage values can also be input from a file.

8.2 Tool Palette

Since the flood plain analysis options are found in the TINs module, the tools are the same as defined in the *TINs* chapter.

8.3 Flood Plain Display Options

The display options dialog box accessed from the *Flood* menu allows control over the display of entities related to drainage analysis.

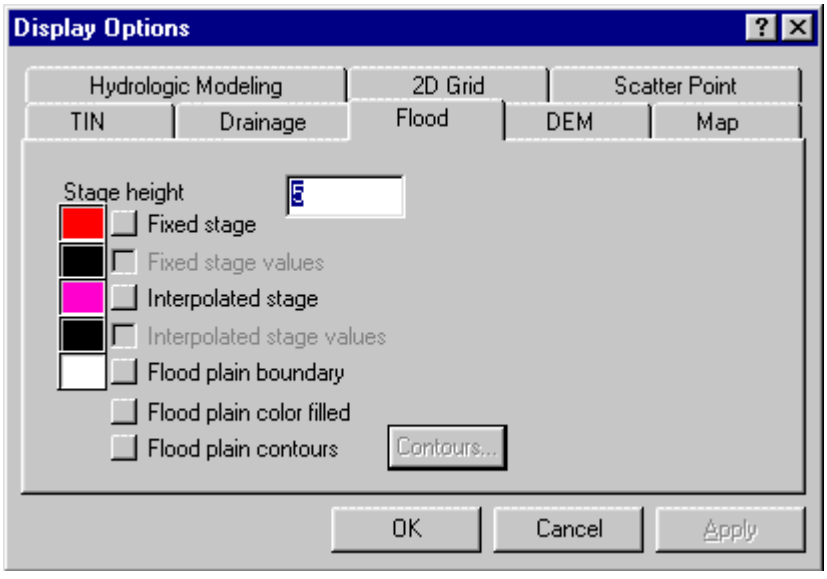


Figure 8-1 Flood Display Options Dialog.

8.3.1 Stage Height

When fixed or interpolated stages are displayed, they are represented as a vertical bar above the vertex. The height of the bar can be controlled by changing the stage height value, which is used as a multiplier.

8.3.2 Fixed Stage

If the *Fixed Stage* option is set, all vertices with fixed stage values will be marked by a colored bar drawn vertically from the vertex. The height of the bar can be adjusted by changing the number of pixels per unit of stage in the *Stage Height* text field. The color of the bar is shown in the color box and can be altered. See the section titled Stage to learn more about fixed and interpolated stage values.

8.3.3 Fixed Stage Values

If the *Fixed Stage Values* option is set, the numerical values of all vertices with fixed stage values will be displayed. The values are displayed above each vertex or above the stage value bars, if they are also displayed.

8.3.4 Interpolated Stage

If the *Interpolated Stage* option is set, all vertices with interpolated stage values will be marked by a colored bar, determined by the color box to the left of the item, and drawn vertically from the vertex. The height of the bar is determined by the value in the *Stage Height* text field where the number of pixels per unit of stage can be specified. See the section on stage to learn more about interpolated stage values.

8.3.5 Interpolated Stage Values

If the *Interpolated Stage Values* option is set, the numerical values of all vertices with interpolated stage values will be displayed. The values are displayed above each vertex or above the stage value bars if they are also displayed.

8.3.6 Flood Plain Boundary

If the *Flood Plain Boundary* option is set, the current flood plain boundary is displayed for the TIN. The color of the flood plain is determined by the color box to the left of the *Flood plain boundary* item.

8.3.7 Flood Plain Color Filled

This option allows you to display the flood plain as a filled polygon.

8.3.8 Flood Plain Contours

If the *Flood plain contours* option is set the flooded region will be contoured according to the depth of the flood. The *Contour Options* in the *Display* menu can be used to control how the contours look, i.e. color filled, isolines, etc.

8.4 Stage

Stage is defined as the depth of water at a given location on a TIN. Depths are represented numerically as absolute values above the present Z value of any vertex on a TIN. For example, a stage value of 3.30 would indicate that the water level would be found 3.30 feet above the elevation of that point. However, when reading stage values from a file or writing them to a file, it is possible to represent the stage values as water surface elevations. Water surface elevations are the numerical sum of the elevation and the stage value at a TIN vertex.

By default, all vertices have a stage value that is equivalent to "unknown" rather than a value of zero which is a valid stage. Stage values are classified by *WMS* as either "fixed" or "interpolated". Fixed stage values are those read from a stage data file or entered interactively by the user. Interpolated stage values are those that are calculated between fixed stage values along stream segments.

8.4.1 Read Stage

When this command is selected from the Flood menu, a file browser is available to find and retrieve the desired flood stage file. The selected file will be opened and read by *WMS*. If the specified file is not a stage file *WMS* will display an error message. The format of the stage file is defined in Appendix A.

Stage values are preserved if multiple stage files are read in. Only those vertices closest to the XY coordinates in a file are assigned stage values. All other TIN vertices retain their previous stage values.

When stage values are defined, a display option that draws a "stage bar" or vertical line at that node, relative in length to the stage value, is automatically turned on. The number of pixels drawn for each increment of stage value can be set by the user in the *Flood* tab of the *Display Options* dialog box. In addition, the option to turn on a numerical display of the stage values when the

stage bar display is turned off, is found in the *Flood* tab of the *Display Options* dialog.

8.4.2 Edit Stage

WMS allows the user to interactively change or assign a stage values to a selected vertex on a TIN with the *Edit Stage* command. If more than one vertex is selected, the stage value entered will be applied to all the vertices selected.

All real number values are acceptable stage values. This allows the user flexibility in manually editing the resulting flood plain created by the delineation. Because the delineation uses a distance weighted interpolation scheme to find the flood plain boundaries, negative stage values can be used to reduce the stage values near that point. This can be used to prevent the flood plain from propagating to areas of the TIN that are known to be unaffected by the flood or are outside the area of study. For example, by using the *Edit Stage* command to enter negative stage values along a ridge, can prevent the flood plain from "spilling over" into another basin that is outside the area of interest.

8.4.3 Remove Stage

This command can be used to remove stage values at any number of vertices. The stage values for all selected vertices will be removed when this command is issued from the Flood menu. If no vertices are selected, the program will ask if stage values for all vertices are to be removed. When stage values are removed, the program sets the stage to an equivalent of "none" rather than to zero, because zero could be a valid stage value.

8.4.4 Interpolate Stage

When this command is selected from the Flood menu, the program will search all stream networks for stream nodes that have "fixed" stage values (defined either from a stage file or from interactive editing). If two stream nodes with stage values are found, all stream vertices between these vertices along the path of the stream will then be interpolated linearly based upon the two known values and the relative distances from them. Stage values will *only* be interpolated in this manner along streams between stream nodes with "fixed" stage values.

This command can be useful to define stage values at all stream nodes without entering them all individually with the *Edit Stage* command. With the *Interpolate Stage* command, stage values could be entered at relatively few stream node locations and then interpolated between these values.

This command allows the user to know which stage values will be used in the calculation of the gradient planes at stream node locations. This interpolation of stream node stage values occurs automatically each time the *Delineate Flood* command is issued. Interpolated stage values are automatically displayed by use of the stage bars like those used to display the fixed or defined stage values.

8.4.5 Adding Stage Streams

The *Create Streams* command found in the *Streams* menu allows the user to delineate flood plains from gradient planes, calculated from the terrain surface, without having to create an entire stream network. This tool is useful for situations when a stream-based gradient plane interpolation of the water surface is desired, but the editing of the TIN required for *WMS* to create the stream is not needed.

Normal creation of stream networks for a TIN requires manual editing in order to topographically represent all necessary channel edges. However, if the only purpose for creating a stream network on a TIN is for flood delineation and the approximate location of the stream is known or can be surmised through visual inspection of the TIN itself, the user can manually enter the stream.

To enter a stage stream, the *Select Strings* tool must be the active tool *(see Select Strings tool* *)* and there must be at least one string selected. The *Create Streams* command will create an outlet point at the first vertex of each string selected unless the first vertex selected is already a member of a stream network, in which case a branch point will be created. A stream will be created that connects the rest of the selected vertices to the outlet or branch point in the order they were selected. However none of the remaining vertices in the selected vertices string may be a member of an existing stream network, as this would create "crossing" streams. Stage streams may be added to existing stream networks regardless of whether the existing streams were added as stage streams or created in the normal fashion.

8.4.6 Save Stage

This command will save the present stage values to a stage data file as described in the section titled Read Stage. Upon selecting this command from the Flood menu, a file browser will be available to change directories and specify a filename. The current working directory will be the last directory written to or read from. Below the file browser, a text field is available to specify a filename for the saved stage file.

After a filename is entered, another dialog box will appear, asking the user to choose the type of stage data file to write. The choices are as described in the

section titled Read Stage: stage value or water surface elevation. The files will be of the same format as shown in Appendix A for stage files.

8.5 Flood Plain Options

WMS offers several options for defining flood plain boundaries. While these parameters do affect the appearance of the flood plain, they differ from the options discussed in the section titled *Flood Display Options*. The flood plain must be delineated for these options to take effect.

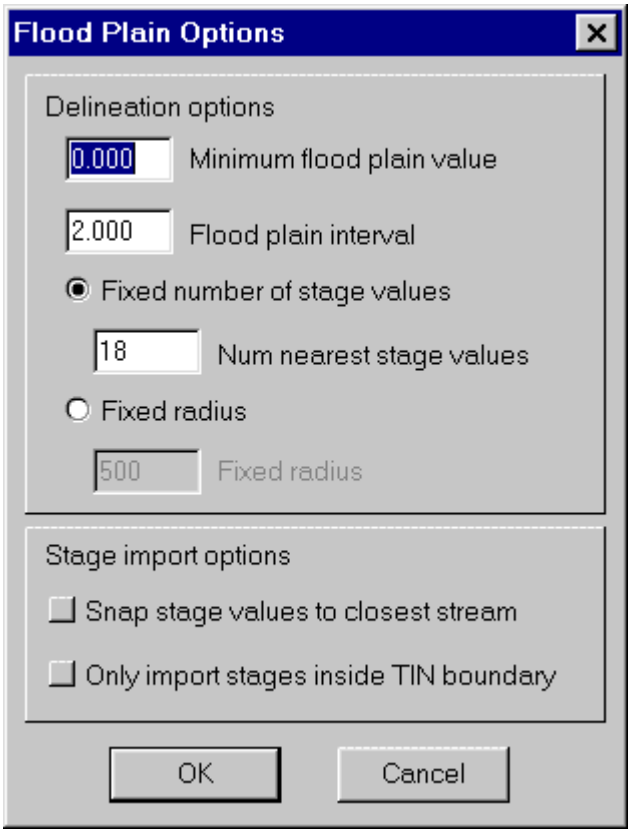


Figure 8-2 Flood Plain Options Dialog.

8.5.1 Minimum Flood Plain Value

The value entered in this data entry indicates the water depth at which the flood plain boundaries will begin to be drawn. Flood plain boundaries are delineated at incremental water depths (see the section titled Flood Plain Interval), with the first boundary being defined at the minimum flood plain value. By default, this value is zero, representing the maximum extent of inundation of the floodwaters. No negative values may be entered for the minimum flood plain value.

8.5.2 Flood Plain Interval

WMS allows the user to delineate flood plain boundaries for multiple water depths simultaneously. Water depths for which flood plains are delineated are determined from the minimum flood plain value and the flood plain interval. The first, or outermost, flood plain is delineated for the water depth specified by the minimum flood plain value. Subsequent flood plains are delineated for incremental water depths, as specified by the flood plain interval, until the maximum water depth is reached. This creates the effect of "flood plain contours", since the multiple flood plain delineations indicate the contours of water depth in the flood plain. If a single flood plain is desired, enter a zero for the flood plain interval and only the flood plain for the minimum flood plain value will be delineated. Only non-negative numbers may be entered as flood plain intervals.

8.5.3 Number of Nearest Vertices with Stage

As described earlier, *WMS* delineates flood plains by means of gradient planes passed through each vertex where a stage value is defined. When calculating the stage contributions of the nearest vertices, *WMS* must know how many "nearest" vertices to use. The default value is 20 but can be changed in this dialog. Since an inverse distance weighted (IDW) function is used to calculate the relative contribution of vertices, using large values here will not change the flood plain definition dramatically. Vertices relatively close to the point in question affect the value of stage at that point much more than those vertices which are relatively far away.

As an alternative to specifying a fixed number of stage vertices, a fixed radius of influence can be set.

8.5.4 Stage Import Options

When importing stage values from a file WMS will find a TIN vertex with an identical xy value as the stage being imported. If you turn on the snap stage values to closest stream option then the stage will be defined at the closest stream node and it does not need to have a matching xy value. You may also specify to eliminate stage values that are completely outside of the TIN boundary.

8.6 Floodplains

WMS delineates flood plains based upon stage values entered by the user interactively, read from a stage data file, or interpolated along a stream between known stage values. The delineation technique makes use of a inverse distance weighted interpolation scheme and gradient plane information calculated for defined stage locations on the TIN as described earlier.

8.6.1 Delineate Flood

This menu selection will delineate a flood plain based upon all fixed and interpolated stage values that are defined on the TIN. As described earlier, *WMS* delineates flood plains by means of gradient planes passed through each vertex where a stage value is defined. When calculating the stage contributions of the nearest vertices, the program must know how many "nearest" vertices to use. This value can be set in the *Flood Plain Options* dialog. The value used will be the lesser of the maximum number of vertices with stage values defined and will be set in the *Flood Plain Options* dialog. The number entered here determines how many vertices are to be used in the delineation interpolation scheme.

A TIN may have only one flood plain delineation associated with it at a time. Therefore, if there is a flood plain currently associated with the TIN, it is replaced with the new flood plain created by the delineation when this command is selected. However, the display of any previous flood plains is not erased. Thus, if a flood plain is currently displayed on a TIN and stage values are altered and a new delineation selected, the new flood plain will be displayed "on top" of the old flood plain. This could be useful in determining the optimum parameters for flood plain delineation. Refreshing the display will cause any "old" flood plains to disappear.

8.6.2 Exporting to a GIS coverage

It is often the case that you would like to store your flood plain boundary in a GIS and therefore want to export from WMS in a shape file format. Since the flood plain is really stored in WMS as a contour line (the contour of zero stage) it cannot be readily converted to feature objects where it could be exported as a shape file (see section 2.8.7).

One way to get around this problem is to create a generic coverage and then digitize around the flood plain boundary and export the feature objects as a shape file. This will work for small flood plains, but will become tedious for larger ones (all things considered though using WMS in this fashion will still be more efficient than trying to do the same thing with other procedures). For more complicated boundaries you may want to use the following work-around:

1. Turn off all of the display options except the flood plain boundary (do have turn on the filled or contour options for the flood plain boundary).

2. Export the flood plain boundary as a DXF file (see page 2-22).

3. Import the DXF file just exported (see page 2-20). You will now have the flood plain in WMS as DXF data.

4. Convert the DXF data to feature objects.

5. You may want to edit the feature objects some more and build polygons.

6. Export the feature objects as a shape file (see page 2-21).

8.6.3 Delete Flood Plain

The current TIN may have only one flood plain associated with it. The current flood plain can be deleted by using the *Delete Flood Plain* command in the Flood menu. When the flood plain is deleted, it is deleted from memory and the display of the TIN is updated. The display of the flood plain in *Display Options* dialog is also turned off.

CHAPTER 9

Topological Trees

9.1 Introduction

A topological tree is one way to represent a watershed in the absence of terrain data. When a digital terrain model or feature objects are being used to automate basin delineation and build a particular sub basin configuration, a tree is simultaneously generated. The tree representation uses icons for confluences (outlet points) and basins, and should be clear to both experienced and inexperienced users of hydrologic models such as *HEC-1* or *TR-20*.

The tree can also be used to create a watershed configuration in the absence of digital terrain or feature object data. The tree representation can then be used to select basins and outlet points for entry of all basin and routing data, in the same way the digital terrain based watershed map can be used. Geometric parameters such as areas, lengths, and slopes, which can be automatically computed when a digital terrain model is used, must be entered interactively. However, all other features of the hydrologic modeling interfaces work in the same fashion.

This chapter describes the functions which can be used to create a topological tree watershed representation when terrain data does not exist. All of the functions for building and/or editing trees cannot be used when a digital terrain model is used. In this case, all new basins and outlet points must be created from the digital terrain model.

9.2 Tool Palette

The following tools are in the dynamic portion of the *Tool Palette*. These tools are available when the *Tree* module is activated. Only one tool is active at any given time. The action that takes place when the user clicks in the *Graphics Window* with the cursor depends on the current tool. The tools are for selection and interactive editing of a topologic tree.

9.2.1 Select Outlets

The *Select Outlets* tool is used to select outlets for operations such as assigning routing or diversion data, creation of new outlets and basins, or deletion.

9.2.2 Select Basins

The *Select Basins* tool is used to select basins for operations such as assigning loss, unit hydrograph, precipitation, and other basin data, as well as deletion. It can be used to select basins from either the TIN or Tree representation of the watershed and behaves identically to the *Select Basin* tools from the *TIN* module.

9.2.3 Select Diversion

The *Select Diversions* tool is used to select diversions for entering/editing diversion data or deletion. Diversions are displayed on the topologic tree only.

9.2.4 Select Hydrographs

The *Select Hydrographs* tool is used to select hydrographs which can then be displayed in the *Hydrograph Window*. Multi-selection operations are available with this tool so that hydrographs from different locations can be overlaid.

9.3 Tree Display Options

The *Hydrologic Modeling* tab in the *Display Options* dialog (Figure 9-1) allows you to control the appearance of items displayed on the topologic tree. A drop-down menu allows you to select when you want the tree to appear on your model: 1) Only when in the Hydrologic Modeling model 2) Always – the tree will always be displayed in the Graphics Window 3) Never – the display of the tree is turned off. Variables can be set to control whether or not the names and icons of the outlet points and basins are displayed.

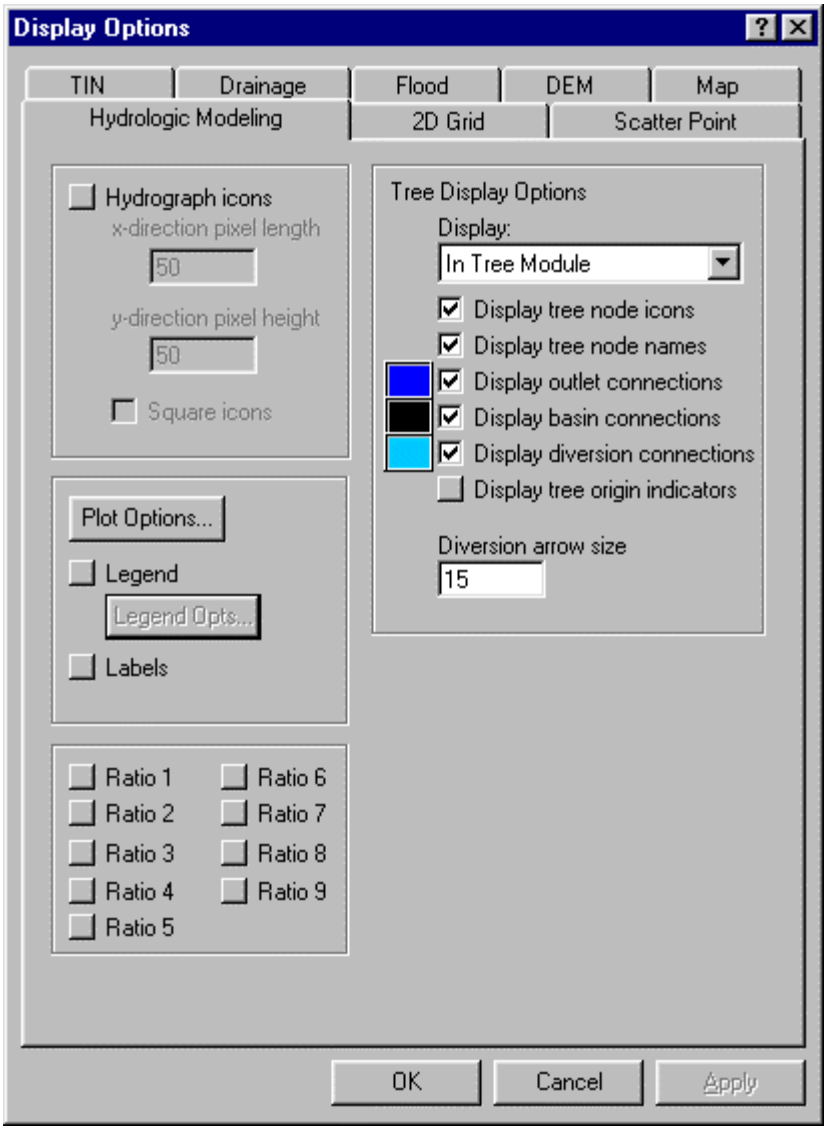


Figure 9-1 Tree Display Options Dialog.

9.3.1 Frame Tree

The *Frame Tree* command is used to center the entire tree within the *Graphics Window*. This is particularly useful after panning and zooming operations.

9.3.2 Collapsing the Topologic Tree

The *Expand Tree Nodes* and *Collapse Tree Nodes* commands in the *Tree* menu allow you to simplify the display of larger, more complicated trees. By selecting an outlet point and collapsing the tree, everything "upstream" from the selected outlet will not be displayed until the same outlet point is selected and the *Expand Tree Nodes* command chosen. A collapsed tree node will be

represented by outlining and connecting it to the rest of the tree with a dashed line. Even though upstream outlets and basins are not displayed, they are still a part of the analysis and would be written to the particular hydrologic model file.

9.4 Default Model

The commands in the *Tree* menu are used in defining any of the hydrologic models supported by *WMS*. The *Default Model* command is used to specify which model should be chosen whenever an operation that requires a choice between one of the hydrologic model is executed. For example double-clicking on a basin or outlet causes the *Edit Parameters* dialog of the default model to come up. The default model is automatically set whenever a file of the given type is read, or if its *Edit Parameters* dialog is accessed from the normal menu commands.

9.5 Creating Trees

In the absence of a digital terrain model, topologic tree representations of watersheds can be created using the commands found in the *Tree* menu. Trees are automatically created when reading existing hydrologic model files.

9.5.1 Add Outlet

The *Add Outlet* command allows the user to enter a new outlet "upstream" from the currently selected outlet node. This represents a new confluence or sub basin outlet which is upstream from the selected outlet. In this fashion, a network of outlet points can be built which actually represent the stream network of the watershed. The *Select Vertex* tool (the same tool used to select outlet points on TINs) can be used to select the outlet points.

9.5.2 Insert Outlet

The *Insert Outlet* command allows the user to enter a new outlet between two selected outlet points. This makes it possible to insert a confluence which may have been inadvertently left out or to simply subdivide a watershed into smaller sub basins. The *Insert Outlet* command can also be used to insert a new outlet at the base of a tree. In such cases then only the bottom most outlet should be selected before issuing the *Insert Outlet* command.

9.5.3 Delete Outlet

The *Delete Outlet* command allows the user to delete selected outlet points. The *Select Vertices* tool can be used to select outlet points which are to be deleted.

9.5.4 Add Basin

The *Add Basin* command defines a new drainage basin for the selected outlet point. Any number of sub basins can be associated with each outlet point. Typically, there are either one or two (a sub basin for each upstream branch), depending on whether or not the outlet point represents a branch in a stream network.

9.5.5 Delete Basin

The *Delete Basin* command can be used to delete a selected drainage basin from a given outlet. The *Select Drainage Basin* tool should be active to select a basin icon.

9.5.6 Add Reservoir

A reservoir can be created for an outlet by selecting the outlet and choosing the *Add Reservoir* command from the *Tree* menu. By creating a reservoir at an outlet storage routing for the outlet can be done followed by routing from the outlet of the reservoir to the next downstream outlet point.

9.5.7 Delete Reservoir

The *Delete Reservoir* command can be used to delete a selected reservoir from a given outlet. The *Select Outlets* tool should be active to select an outlet icon where a reservoir has been defined from the *Tree* menu.

9.5.8 Add Diversion

Diversions are defined in a two step process: by defining the diversion at the outlet or basin where flow is to be diverted, and then assigning a previously defined diversion (retrieving a diversion) to another outlet. Diversions are defined by first selecting an outlet point or a drainage basin where the flow is diverted and then selecting the *Add Diversion* command from the *Tree* menu.

After adding a diversion, a link will be drawn from the outlet or basin selected to the side of the *Graphics Window*, indicating that no retrieval outlet has yet been specified for that diversion.

The retrieval outlet point can be specified when defining diversions from one outlet to another by multi-selecting the retrieval outlet before issuing the *Add Diversion* command.

9.5.9 Retrieve Diversion

To retrieve a diversion at an outlet point, select the outlet point where the diverted flow is to be retrieved and then select the *Retrieve Diversion* command from the *Tree* menu. If more than one diversion not yet retrieved exists, a list of defined diversions without retrieval outlet points (a diversion cannot be assigned to more than one retrieval outlet) will appear in the diversion selection dialog. Select the name of the desired diversion and choose OK.

After retrieving a diversion, the diversion link will now be drawn from the outflow outlet or basin to the retrieval outlet point. *HEC-1* only allows inflow to be defined at outlet points.

9.5.10 Delete Diversion

The *Delete Diversion* command can be used to delete a selected diversion. The *Select Diversions* tool should be active to select a diversion icon.

9.6 Viewing Hydrographs

Hydrographs can be read into *WMS* and displayed in icon form at the appropriate basin or outlet.

For an *HEC-1* analysis, a TAPE22 file can be read into *WMS*. Any number of hydrograph sets (TAPE22 files) may be read into *WMS* and displayed in the *Hydrograph* window. The name of the TAPE22 file is given as the solution file when running *HEC-1* from the *HEC-1* menu.

For a *TR-20* analysis the GRAPHICS file, which stores discharge hydrographs, can be read into *WMS*. This options is only available when running the version of *TR-20* distributed with *WMS*. This file is specified as part of the "all other files" prefix when running *TR-20* from within *WMS*, and always has the three letter extension of THY.

For the NFF and rational analysis programs hydrographs can be computed from dimensionless unit hydrographs applied to the computed peak discharges. These hydrographs are automatically stored and displayed.

Hydrographs are displayed in the *Hydrograph Window* by using the *Select Hydrograph* tool and choosing the desired hydrographs. Multiple hydrographs may be selected and overlaid in the window at the same time, and several

different display options can be used while examining hydrographs. See section 9.6.4 for viewing hydrographs in tabular form.

By default the *Hydrograph Window* is hidden. It can be activated by selecting the *Show Hydrograph Window* command in the *Display* menu. Once activated in can be hidden again by selecting the *Hide Hydrograph Window* command in the *Display* menu.

9.6.1 Display Options

The *Hydrologic Modeling* tab in the *Display Options* (see Figure 9-1) allows control over the appearance of hydrographs in the *Hydrograph Window*. Each set of hydrographs read into *WMS* is assigned a color and a set name. Hydrographs of the same set will all be displayed using the same color. The legend displays all set names in their appropriate colors. The legend options can be used to change set names and/or colors.

Labels to identify the outlet or basin associated with the hydrograph can be displayed. An X and Y grid can be turned on for closer examination of time to peak, max flow, etc. Options controlling the display of these two grid axes can be specified. A major and minor title can be entered and displayed at the top of the hydrograph window.

The size of the icons used to display hydrographs on the TIN and tree can be specified in the appropriate edit fields.

If multiple ratios of a storm have been read, then the display of certain hydrograph ratios can be specified.

The contents of the hydrograph window can be output to a printer (PostScript file on UNIX computers) using the *Print* command from the *File* menu.

9.6.2 Reading Hydrographs

Hydrographs can be read into *WMS* from a results files using the *Open* command in the *Hydrographs* menu. Hydrographs are generated for each basin and outlet point when running one of the hydrologic models. Two hydrographs may be generated for the outlet points, one representing the combination of "upstream" hydrographs, and one representing a hydrograph which has been routed to the next downstream outlet point.

If multiple storms or multiple ratios of a given storm are analyzed in the same run of *HEC-1*, then multiple hydrographs for each basin and outlet point will computed. If multiple storms have been defined using the JD card, the index numbers of hydrographs to be read must be specified. All indices or particular numbers can be specified. If multiple ratios of the same storm exist (defined on JR records), all ratios will be read. The display of a given ratio is

controlled from within the *Hydrologic Modeling* tab in the *Display Options* dialog. Currently multi-plan storm hydrographs are not read.

Each storm index becomes a separate hydrograph set. Hydrograph sets are named by appending the index number to the file from which it is read. While it may be convenient to read all indices each time, it may produce an overwhelming number of hydrographs and make the display too cluttered for comparisons.

It is often useful to read a hydrograph not computed by *HEC-1*, or one of the other hydrologic models for calibrating purposes. This can be done by creating a hydrograph file using the *WMS* file format. The file format is given in chapter 20.

9.6.3 Deleting Hydrograph Sets

All hydrographs read into *WMS* can be deleted at once using the *Delete All* command in the *Hydrographs* menu. The *Delete Previous* command can be used to remove the most recent set. Deleting the previous hydrograph set when it is no longer needed for display and comparison eliminates the confusion created by displaying too many hydrographs in the *Hydrograph Window*.

9.6.4 List Hydrographs

Hydrographs values can be listed in a tabular format using the *List* command in the *Hydrograph* menu. All currently selected hydrographs will be listed in a table in this dialog. You can output the hydrograph to either a *WMS* formatted hydrograph file, or a standard spreadsheet formatted file (tab, comma, or space delimited) from within this dialog. The dialog is shown in Figure 9-2.

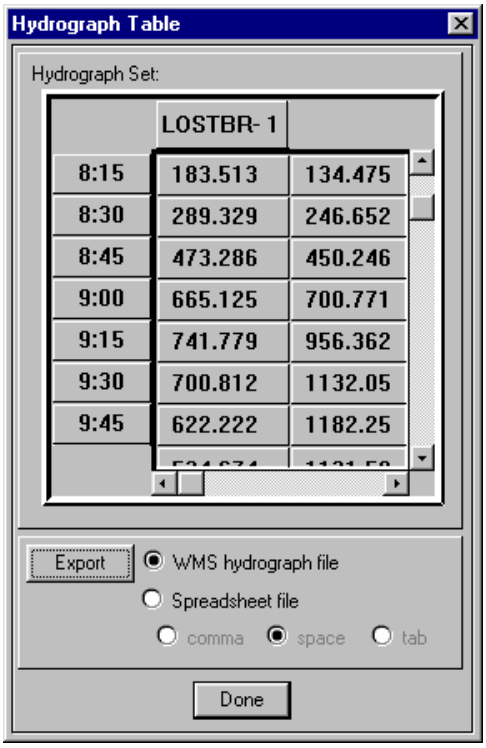


Figure 9-2 Hydrograph List Dialog.

9.7 WMS Tree File

WMS supports several different hydrologic models. Unfortunately, all the tree data stored in *WMS* cannot be stored in any single model file. For example, create an HEC-1 simulation and an NFF simulation in *WMS*, then save an HEC-1 or NFF file to your computer's hard disk. If you save only an HEC-1 file, close *WMS*, start it up later, then read the HEC-1 file in, you will lose the NFF data you entered into *WMS*. Alternatively, if you save an NFF file, close *WMS*, start it up later, then read in the NFF file, you will lose your HEC-1 data.

Some models, such as the Rational method and NFF, do not support diversions. Therefore, if a diversion is created in one of these models, it is not saved when the model file is saved.

To fix these problems, the *WMS* Tree file has been created. The *WMS* Tree file contains all the data necessary to restore all the tree data from all the models in *WMS*. When creating a model file, you will need to save out a model file when running the model from *WMS*. But you should save all your tree data to a *WMS* tree file as well. Doing this will insure that all your tree data will be restored each time you re-start *WMS*.

To save a Tree file, simply select the *Save As…* command from the *File* menu. This will give you all the options for saving a super (project) file. Make sure you are saving out a Tree file; specify a filename for the Tree file, and the Tree file will be saved. You can read a Tree file in from the *Open* command in the *File* menu.

CHAPTER 10

HEC-1 *Interface*

10.1 Introduction

In order to provide an integrated hydrologic analysis environment, the *HEC-1* model can be defined using the commands in the *HEC-1* menu and resulting dialogs. There are six steps in defining an *HEC-1* model using *WMS*:

10.1.1 Create a Topologic Tree

In the absence of terrain data this is done using the commands in the *Tree* menu as described in the previous chapter. When using a terrain model to define a watershed and sub-basin boundaries a topologic tree is automatically created. Diversions can not be created directly on a terrain model, but must be created using the commands in the *Tree* menu.

10.1.2 Define Job Control Parameters

Job control parameters are used to define the time and length of a simulation, output diagnostic controls, and other miscellaneous items. Definition of these parameters is discussed in detail later in this chapter.

10.1.3 Edit Basin/Outlet/Reservoir/Diversion Data

Parameters for all hydrograph stations are entered/edited using the *Edit HEC-1 Parameters* dialog accessed from the *HEC-1* menu. This dialog displays

current values and lets you edit values for the currently selected hydrograph station. More details about using this dialog are given later in this chapter.

If gages are used as part of the precipitation method, they should be defined sometime during this step. Gages can be set up using a feature object coverage (see page 3-21), or by entering the coordinates directly in the gage dialog (see page 10-32).

10.1.4 Model Check

This step allows you to check all *HEC-1* data prior to performing an analysis. It reports errors such as zero area, undefined losses, etc. These errors should be corrected by returning to previous steps before actually trying to run *HEC-1*.

10.1.5 Run *HEC-1*

WMS creates *HEC-1* files compatible with any version of *HEC-1*. However, a Windows (X-Windows on UNIX and MS-Windows on PC's) compiled version of *HEC-1* is distributed with *WMS* so that it can be run without leaving the *WMS* environment. This command is explained in more detail later in this chapter.

10.1.6 View Hydrographs

Once an *HEC-1* simulation has been run you can view resulting hydrographs using the commands described in the previous chapter. After viewing the hydrographs you may wish to repeat the previous steps in order to calibrate a model, or look at different scenarios.

The remainder of this chapter describes how data for creating an *HEC-1* is entered using *WMS*. For an in depth description of the *HEC-1* runoff model itself, refer to the *HEC-1* User's Manual.

10.2 Tool Palette

The tools used for selecting outlets, basins, and diversions are defined in the other modules (*Hydrologic Modeling, Map, DEM*, and *TINs*). When entering *HEC-1* data for basins or outlets defined parameters will be assigned to all currently selected basins or outlets.

10.3 Job Control

The *Job Control* dialog (Figure 10-1) is used to define general information about the *HEC-1* model. Selecting the Job Control command from the HEC-1 menu accesses this dialog. A list of the data entries for each item in the *Job Control* dialog with a short description and the two letter *HEC-1* card abbreviation for each follows. The *Job Control* dialog can also be accessed by toggling on the display of *Job Control* cards in the *Edit HEC-1 Parameters* dialog and then clicking on a *Job Control* related card in the text display window.

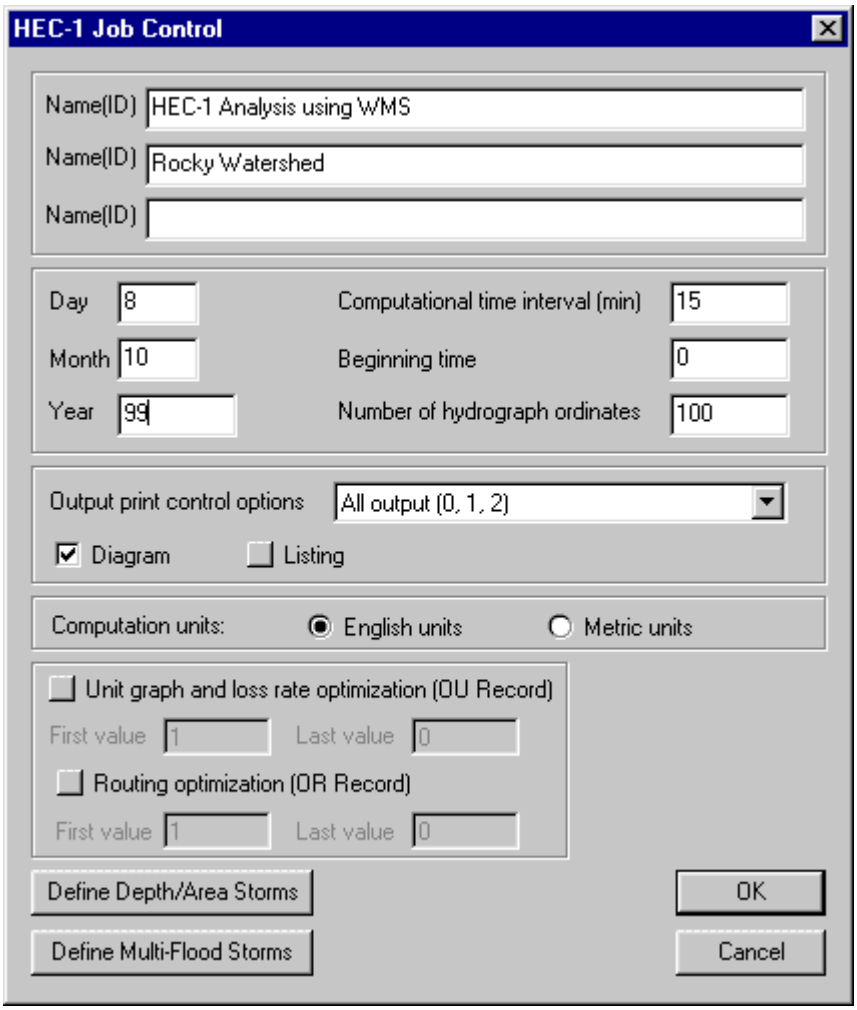


Figure 10-1 HEC-1 *Job Control Dialog.*

10.3.1 Name (ID)

Enter a name and/or project description identifying the model. Three different name records up to 78 characters each can be entered. The name records will appear at the top of the *HEC-1* input file.

10.3.2 Day, Month, Year (IT)

The day, month, and year fields correspond to the date of the first computational time interval. The year is entered as the last two digits only. **NOTE: The date should be consistent with dates defined on IN records for precipitation and other time series data.**

10.3.3 Computational Time Interval (IT)

The computational time interval defines the length in time between hydrograph ordinates. The interval should be specified in minutes, with 1 being the minimum. The total simulation time of the model will be the number of ordinates minus one times the computational time interval.

10.3.4 Beginning Time (IT)

The beginning time of the simulation is defined in this entry. The time should be specified as a single integer number defining the hours and minutes. For example 7:45 am would be entered as 745 and 1:15 PM would be 1315. **Do not place a colon between the hours and minutes**.

10.3.5 Number of Ordinates (IT)

The number of hydrograph ordinates computed during the simulation is defined in this entry. The length of the simulation is determined by the number of ordinates times the computational interval. If a simulation is run and a complete hydrograph is not developed, then the number of ordinates can be increased to increase the length of simulation. Similarly the number of ordinates can be decreased if the simulation continues long past the falling limb of the hydrograph.

10.3.6 Output Control Options (IO)

HEC-1 generates an ASCII output file that can be printed or read into any standard word processing program. The information output is controlled by the output control options in this dialog.

10.3.7 Units (IM)

By default *HEC-1* performs computations in English units, however metric unit calculations can be specified.

10.3.8 Tree Diagram (*DIAGRAM)

The diagram option causes a diagram of the stream network to be printed to the *HEC-1* output file. This diagram should appear similar to the tree representation created by *WMS*.

10.3.9 Listing (*LIST)

Causes *HEC-1* to echo print input data.

10.3.10 DEPTH/AREA STORMS (JD)

Runoff simulations which use a consistent depth/area relationship as defined in the *HEC-1* User's Manual can be defined by clicking on the button to bring up the dialog shown in Figure 10-2.

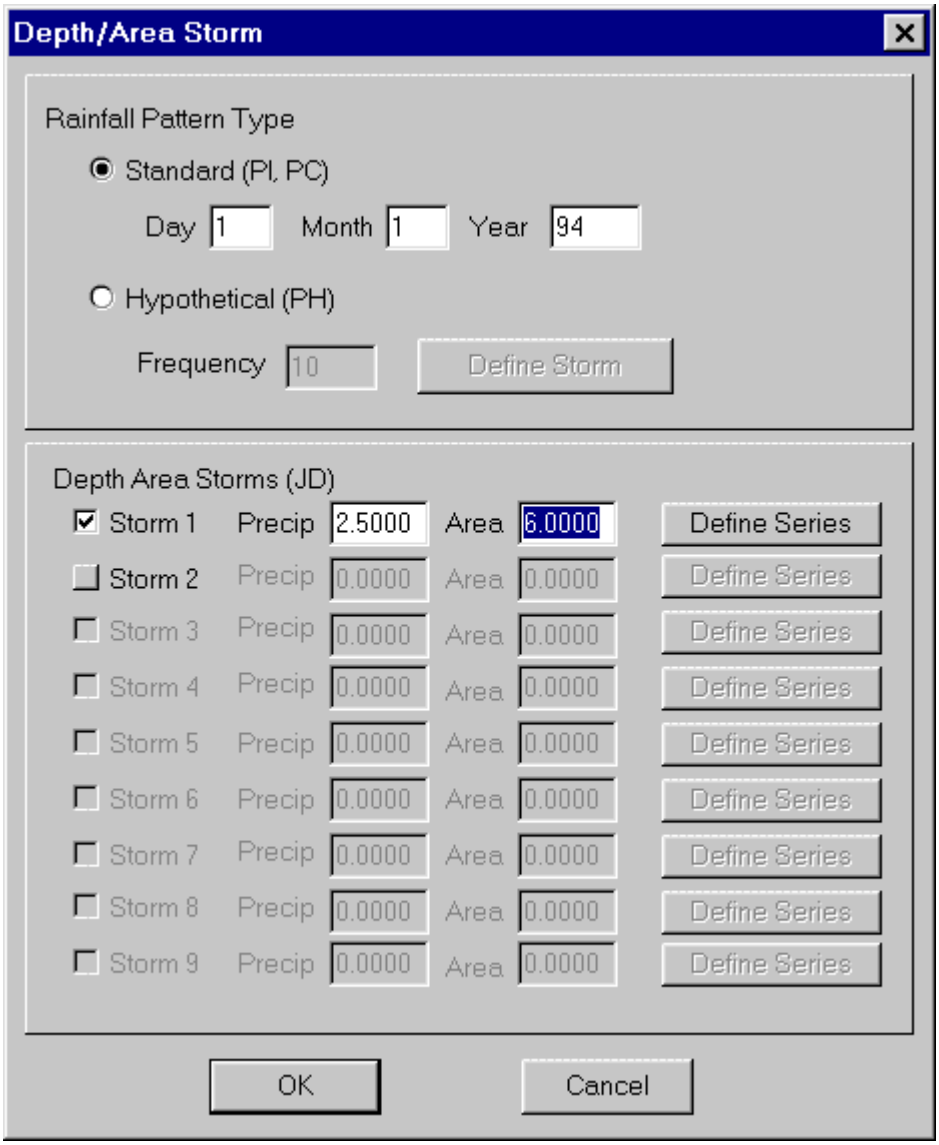

Depth/Area Storm
Rainfall Pattern Type
Standard (PI, PC)
Day 1 Month 1 Year 94
Hypothetical (PH)
Frequency 10
Define Storm
Depth Area Storms (JD)
Storm 1 Precip 2.5000 Area 6.0000 Define Series
Storm 2 Precip 0.0000 Area 0.0000 Define Series
Storm 3 Precip 0.0000 Area 0.0000 Define Series
Storm 4 Precip 0.0000 Area 0.0000 Define Series
Storm 5 Precip 0.0000 Area 0.0000 Define Series
Storm 6 Precip 0.0000 Area 0.0000 Define Series
Storm 7 Precip 0.0000 Area 0.0000 Define Series
Storm 8 Precip 0.0000 Area 0.0000 Define Series
Storm 9 Precip 0.0000 Area 0.0000 Define Series
OK
Cancel

Figure 10-2 Depth/Area Storm Dialog.

Rainfall Pattern Type

Each storm (JD record) may be defined with a set of PI/PC cards giving the precipitation pattern to be used for that depth and area. If no pattern is given for the second through ninth storms then the previously defined precipitation pattern will be used.

Alternatively hypothetical storms may be used to specify precipitation patterns. In such cases only a single storm using the PH record needs to be defined.

The rainfall pattern type is specified using the radio group at the top of dialog. as a new storm is activated, data fields which require input will be undimmed.

Data fields which do not apply to the specified pattern type will remain dimmed.

Depth Area Storms

Different depth/area storms can be defined by toggling on a storm from within the Depth/Area Storms portion of the dialog.

Precipitation

The average precipitation is entered in this data field.

Area

The applicable area for this storm is entered in this data field.

Defining a Series

If a standard storm type has been chosen, a rainfall pattern must be defined using the *XY Series Editor*. Activate the *XY Series Editor* by choosing the *Define Series* button. A series for the first storm must be defined. If a series is not defined for the second through ninth storms, the previously defined storm will be used.

10.3.11 MULTI-FLOOD STORMS (JR)

Multiple ratios of a given storm event can be specified by bringing up the *Multi-flood* dialog (Figure 10-3) using the appropriate button. The data input options are described below.

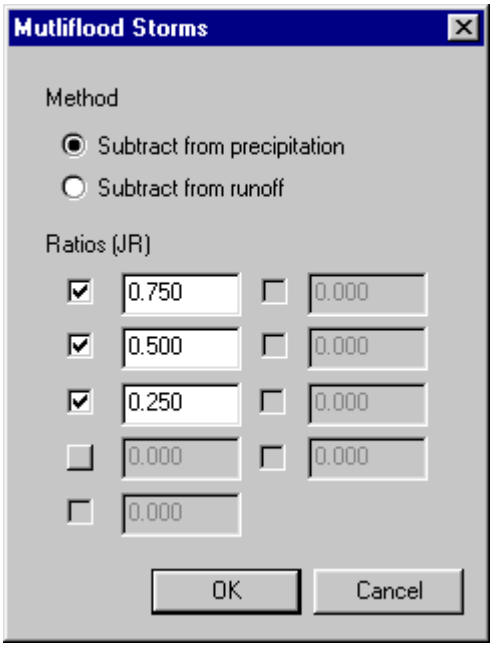


Figure 10-3 Multi-Flood Storm Dialog.

Method

Input ratios can be specified using either precipitation or runoff.

Ratios

Up to nine different storm ratios can be specified by toggling on the appropriate check box. All hydrograph or precipitation ordinates are multiplied by the specified value.

Each storm is analyzed during execution of *HEC-1* and the resulting TAPE22 file will contain as many hydrographs for each station as there are ratios defined. Display of different storm ratios can be toggled on and off in the *Hydrologic Modeling* tab of the *Display Options* dialog.

10.3.12 Unit Graph and Loss Rate Optimization (OU)

Select this option if you want to optimize unit hydrograph and loss rate parameters in this *HEC-1* simulation so the calculated hydrograph will match the observed hydrograph.

10.3.13 Routing Optimization (OR)

Select this option if you want to optimize routing parameters using observed inflow and outflow hydrographs and a pattern lateral inflow hydrograph for the routing reach.

10.4 Entering/Editing HEC1 Parameters

Attributes or parameters for all *HEC-1* hydrograph stations are defined and/or later edited using the *Edit HEC-1 Parameters* dialog. This dialog is accessed by selecting the *Edit HEC-1 Parameters* command from the *HEC-1* menu or by double-clicking on basin, outlet, or diversion icon from the *Graphics Window*. Figure 10-4 shows this dialog.

If a basin, outlet, or diversion is selected before issuing the command then data for that object appears in the text window at the bottom of the dialog. The top portion of the dialog lists the *HEC-1* cards that can be edited by selecting the corresponding button. When a hydrograph station is selected (basins/outlets/diversions) only the buttons that edit parameters associated with that hydrograph station are active, all others are dimmed. In addition to using the appropriate button, *HEC-1* attributes can be edited by clicking on the *HEC-1* card in the text display window. Using this method, job control parameters can be edited by first toggling their display using the *Display Job Control Cards* toggle box, and then selecting a job control card from the text display window.

Once the dialog appears it becomes part of the main screen until you select the Done button. Therefore, you can continue to select additional, or other hydrograph stations so that data for that object may be edited without exiting the dialog. You can use the *previous and next hydrograph station* buttons to cycle through hydrograph stations in the order they are computed by *HEC-1*. Since the dialog is part of the main screen, all menu commands are active while this dialog is present.

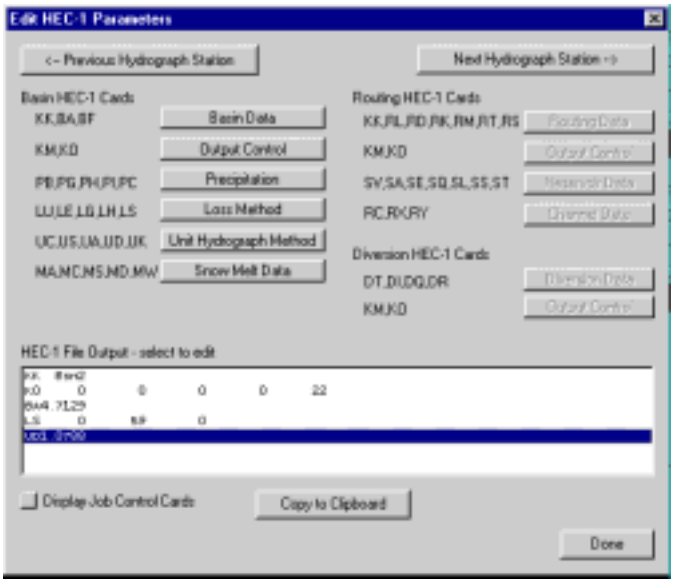


Figure 10-4 Edit HEC-1 *Parameters Dialog.*

The remaining sections in this chapter cover the attributes which can be edited from each of the buttons in the *Edit HEC-1 Parameters* Dialog.

10.5 Output Control(KO)

For each hydrograph station (basin hydrographs, combined hydrographs, and routed hydrographs) different output controls can be specified. This dialog is accessed by selecting the Output Control dialog button from the *Edit HEC-1 Parameters* dialog. Entries which can be defined in this dialog are described below.

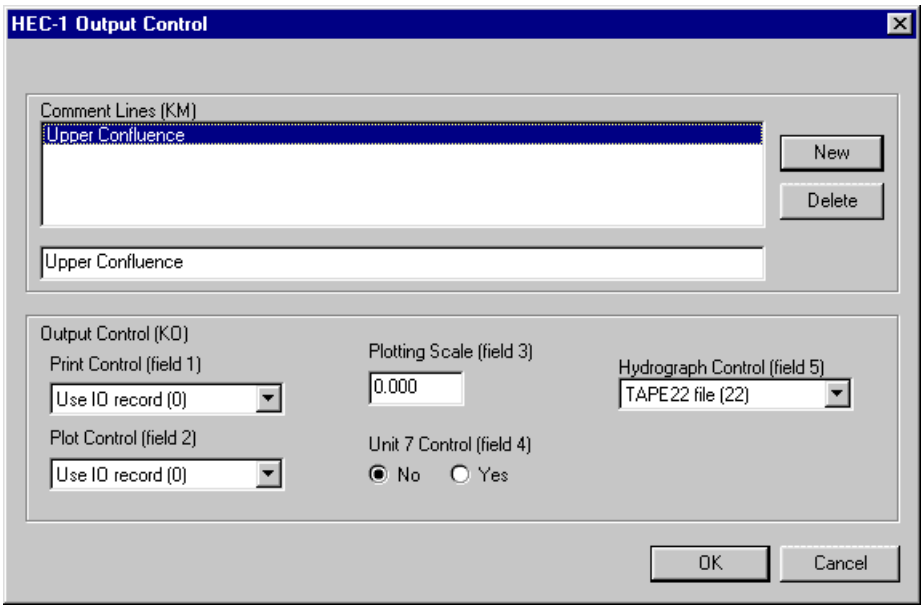


Figure 10-5 HEC-1 *Output Control Dialog.*

10.5.1 Routed and Combined Hydrographs at Outlets

In *WMS* an outlet point is used to represent locations where hydrographs are both combined and then routed. Therefore, if an outlet is selected before choosing the *Output Control* dialog, a radio group at the top of the dialog appears so that you may specify whether the options should be applied to the combined or routed hydrograph. If a basin is selected the radio group at the top of the dialog does not appear.

10.5.2 Comment Lines (KM)

Individual comments can be defined for each hydrograph station. These comments can be used to identify unique characteristics about a particular basin or outlet point. A new comment can be defined by selecting the new button in the *Output Control* dialog and then entering the comment in the text

entry. When more than one comment card has been defined, the up and down arrow buttons can be used to scroll through the list of comments for that hydrograph station. When using *WMS*, comment cards always appear directly after the KK cards for each hydrograph station.

10.5.3 Output Control (KO)

These controls determine what information about a given hydrograph station is written to the *HEC-1* ASCII output file. By default the IO record information is used. However, you may wish to print out a more (or less) complete summary for individual hydrograph stations.

By default, the option to write a hydrograph to the TAPE22 file is specified. This is the file read by *WMS* for display of hydrographs. Therefore, this option should only be changed to suppress particular hydrographs.

10.6 Basin Data

General information for each basin is entered by selecting a basin and choosing the *Basin Data* dialog (Figure 10-6). If multiple basins are selected the appropriate parameters are applied to all. A description of the data entry fields in this dialog is given below.

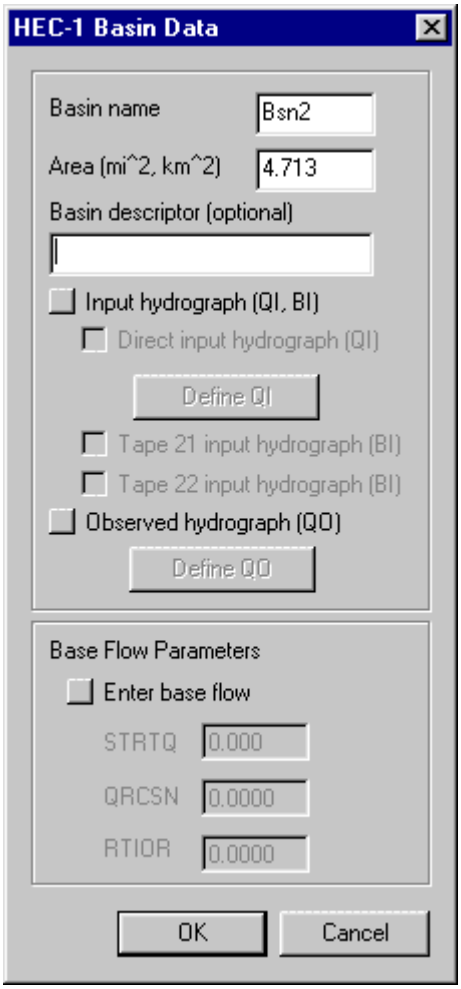


Figure 10-6 HEC-1 *Basin Data Dialog.*

10.6.1 Basin Name (KK)

Each hydrograph station should be identified with a unique name. This name appears as part of the KK record for that station in the input file. The name should not be more than six characters long. By default *WMS* uses the basin ID number followed by a "B" for the name, but a descriptive name is generally more useful.

10.6.2 Basin Area (BA)

When a terrain model is used, basin areas and slopes can be computed automatically using the *Compute Basin Data* command from the *Drainage* menu of the *TIN* or *DEM* module, or the *Update Basin Data* command in the *Feature Objects* menu of the *Map* module. Otherwise, areas and slopes must be entered interactively using the topological tree as a map. Areas should be entered in either square miles or square kilometers.

10.6.3 Direct Input Hydrograph (QI)

If a hydrograph is known for a given basin, there is no need to compute a synthetic hydrograph. This hydrograph can be input by selecting the check box and then defining the hydrograph using the *XY Series Editor*.

Precipitation, base flow, loss rates, and unit hydrograph methods for each hydrograph must be specified, regardless of whether or not a TIN is being used, before a complete *HEC-1* file can be created. Selecting all of the basins enters data for one or more basins. **NOTE: If no basins are selected, the information entered is applied to all basins.**

10.6.4 Observed Hydrograph (QO)

This record is used to input an observed hydrograph for an optimization job (OU record). This hydrograph can be input by selecting the check box and then defining the hydrograph using the *XY Series Editor*.

10.6.5 Base Flow (BF)

Base flow parameters can be defined for a basin by selecting the *Enter base flow* check box. The input parameters for base flow are as follows:

- STRTQ - Flow at the start of the storm in cfs (cms for metric units).
- QRCSN - Flow in cfs (cms) below which base flow recession occurs in accordance with the recession constant RTIOR. In other words, it is that flow where the straight line (in semilog paper) recession deviates from the falling limb of the hydrograph.
- RTIOR - The ratio of the recession flow (QRCSN) to that flow occurring one hour later (Must be greater than or equal to 1).

10.7 Precipitation

Precipitation patterns are assigned to basins by first selecting the appropriate basin(s) and then clicking on the precipitation button in the *Edit HEC-1 Parameters* dialog. If multiple basins are selected then the defined parameters will apply to all selected basins. **NOTE: If no basins are selected, the parameters can be applied to all basins.**

Four different precipitation types can be specified using the dialog shown in Figure 10-7. Their data entry fields are described in the following sections.

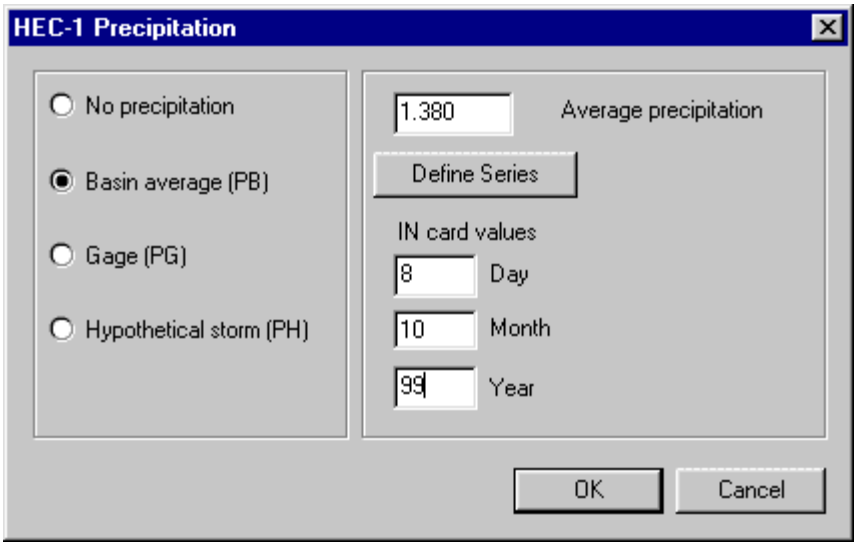


Figure 10-7 HEC-1 *Precipitation Dialog.*

10.7.1 No precipitation

If no precipitation for a given basin is chosen, then the program will use the precipitation pattern of the most recently defined basin. In other words, if the same precipitation pattern is to be used for each basin, specify precipitation at the upper-most basin and let all other basins "inherit" this same pattern.

10.7.2 Basin Average (PB)

With this method, a time distribution can be entered to create a PI or PC card. The distribution is entered via the *XY Series Editor*, refer to the chapter titled Using the *XY Series Editor*. Several standard storm distributions can be loaded automatically from this editor. In addition, distributions can be saved and later restored from a file. When creating PI or PC records, an IN record needs to be defined to specify the beginning time and date of the storm. Be sure that these values are consistent with the time parameters entered on the IT card.

10.7.3 Gage (PG, PT, PW, PR)

Gages can be used with or without a terrain model. If drainage basins have been defined using a TIN, the appropriate gage weights (using the Thiessen polygon method) for each basin are automatically computed when the *Compute (or Update) Basin Data* command is executed. If the *HEC-1* model is defined using only the tree, or to change any of the computed values, the gage weights can be changed/assigned by clicking on the gage weights button in the *Precipitation* dialog. The gage weights dialog will display a list of all defined gages and their station type. Choose from this list when defining gage

weights to the selected basin. Storm total stations are written on PT/PW records whereas recording stations are written on PR/PW records. You must have at least one PR/PW record combination for each basin.

When using a terrain model (DEM, TIN, or Feature Objects) a rain gage coverage can be defined and used to establish the positions of gages by using the graphical creation/selection tools available in the *Map* module. For more information see page 3-21.

10.7.4 Hypothetical Storms (PH)

A hypothetical storm may also be used to define the precipitation pattern for the runoff simulation. The *XY Series Editor* is used to define the necessary rainfall values for the appropriate times. The storm frequency in percent is entered in the frequency edit field. Rainfall will be converted to an annual-series for fifty, twenty, and ten percent storms. No conversion is made for any other frequency storms. A storm area to be used in computing reduction of point rainfall depths is entered in the area edit field. If a 0 is entered for the area then the basin area (or area from JD card for depth/area storms) will be used as a default.

10.8 Loss Methods

One of several different loss methods can be chosen when generating synthetic hydrographs. A loss method is assigned to a basin by first selecting the basin and then choosing the *Loss Method* button in the *Edit HEC-1 Parameters* dialog. As with other basin data the same parameters can be assigned to several basins by selecting multiple basins before accessing the *Loss Method* dialog.

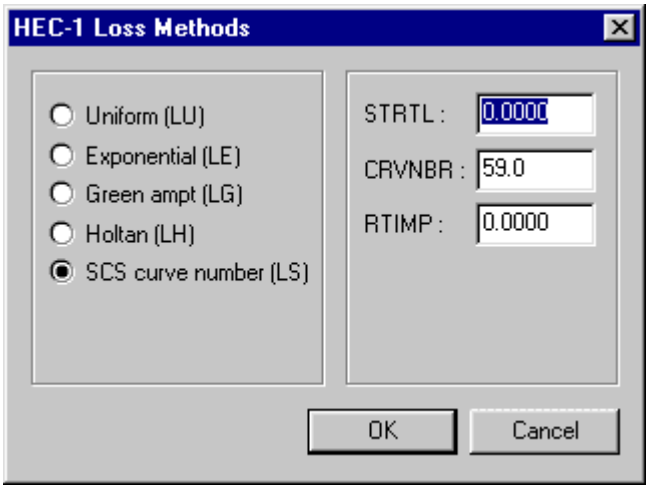


Figure 10-8 HEC-1 *Losses Dialog.*

When defining a kinematic wave model, it may be necessary to define a separate set of loss parameters for the two different UK records (generally corresponding to pervious and impervious area). This second set of loss parameters is defined from within the *Unit Hydrograph Method* dialog.

The available loss methods, along with a description of their important parameters is given below.

10.8.1 Uniform (LU)

This loss method uses an initial value and a uniform value to define infiltration losses. Input parameters are as follows:

- STRTL - Initial rainfall/snow melt loss in inches (mm) for snow free ground.
- CNSTL - Uniform rainfall/ loss in inches/hour (mm/hour) which is used after the starting loss (STRTL) has been satisfied.
- RTIMP - Percentage of drainage basin that is impervious.

10.8.2 Exponential (LE)

Parameters for the exponential loss method are as follows:

- STRKR - The starting value of the loss coefficient on the exponential recession curve for rain losses.
- DLTKR - The amount in inches (mm) of initial accumulated rain loss during which the loss coefficient is increased.
- RTIOL - Parameter computed as the ratio of STRKR to a value of STRKR after ten inches (ten mm) of accumulated loss.
- ERAIN - Exponent of precipitation for rain loss function that reflects the influence of the precipitation rate on basin-average loss characteristics.
- RTIMP - Percentage of drainage basin that is impervious.

10.8.3 Green Ampt (LG)

Green-Ampt infiltration loss parameters are as follows:

- IA - Initial loss (abstraction) in inches (mm).

- DTHETA - Volumetric moisture deficit. If this value is 0, then the method reduces to the initial loss equal to IA and a constant loss equal to XKSAT.

- PSIF - Wetting front suction in inches (mm). If this value is 0, then the method reduces to the initial loss equal to IA and a constant loss equal to XKSAT.

- XKSAT - Hydraulic conductivity at natural saturation in inches/hour (mm/hour).

- RTIMP - Percentage of drainage basin that is impervious.

Using methods defined by the Maricopa County Flood Control District, Green-Ampt parameters can be determined from GIS data layers automatically in WMS. See section 15.2 for information on how to do this.

10.8.4 Holtan (LH)

Parameters used to define the Holtan loss method:

- FC - Holtan's long term equilibrium loss rate in inches/hour (mm/hour) for rainfall/ losses on snow free ground.

- GIA - Infiltration rate in inches/hour per inch**BEXP (mm/hour per mm**BEXP) of available soil moisture storage capacity.

- SAI - Initial depth in inches (mm) of pore space in the surface layer of the soil which is available for storage of infiltrated water.

- BEXP - Exponent of available soil moisture storage.

- RTIMP - Percentage of drainage basin that is impervious.

10.8.5 SCS (LS)

The SCS curve number method uses the following parameters:

- STRTL - Initial rainfall abstraction in inches (mm) for snowfree ground. If value is 0, then initial abstraction will be computed as .2*(1000-10*CRVNBR)/CRVNBR.

- CRVNBR - SCS curve number for rainfall/ losses on snowfree ground. **Note:** Composite Curve Numbers can be computed automatically when this method for computing losses is chosen and a terrain model is present. Refer to section 15.2 for more information on computing curve numbers.

- RTIMP - Percentage of drainage basin that is impervious.

10.9 Unit Hydrograph Methods

One of several different unit hydrograph methods can be chosen when generating synthetic hydrographs. A method is assigned to a basin by first selecting the basin and then choosing the *Unit Hydrograph Method* button from the *Edit HEC-1 Parameters* dialog. As with other basin options the same parameters can be assigned to several basins by selecting multiple basins before accessing the *Unit Hydrograph Method* dialog.

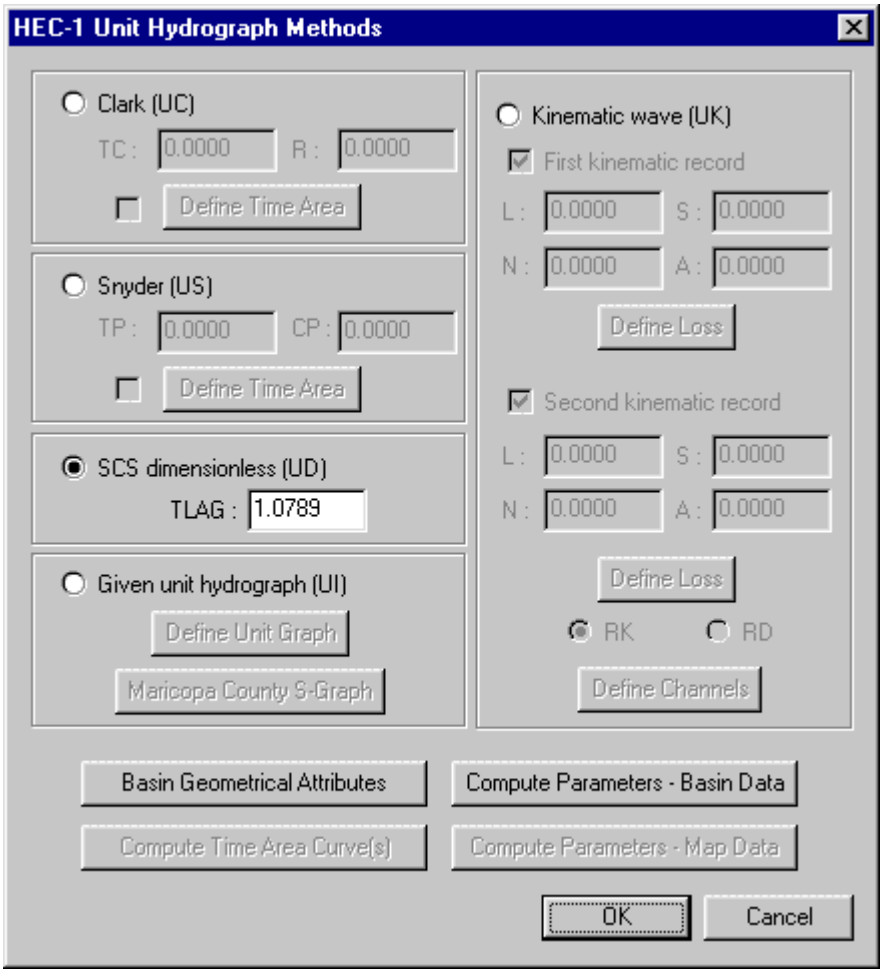


Figure 10-9 HEC-1 *Unit Hydrograph Dialog.*

10.9.1 Clark (UC)

The parameters for the Clark method are as follows:

- TC - Time of concentration in hours for the unit hydrograph. Several different equations exist for determining the time of concentration. The list of basin geometric attributes computed automatically when basins have been delineated from a terrain model can be useful in many of these equations. These attributes can be viewed from within the *Unit Hydrograph Method* dialog by choosing the *View Basin Geometrical Attributes* button. Time of concentration can be computed from one of several equations using these attributes, or by using a time computation coverage. These options are accessed from the *Compute Parameters – Basin Data* and *Compute Parameters – Map Data* buttons respectively (see section 15.3).

- R - The Clark storage coefficient in hours.

- TIME AREA CURVE - The time area curve defines the area of the watershed contributing runoff to the basin outlet as a function of time. This curve is defined by selecting the check box and then activating the *XY Series Editor* with the adjacent button. The time area curve can be computed automatically from a TIN (this method will not work for watersheds delineated from DEMs or Feature Objects) using the *Compute Time Area Curves* button.

10.9.2 Snyder (US)

Parameters for the Snyder unit hydrograph are as follows:

- TP - Lag time in hours. Several different equations have been published to determine the lag time of a basin. Many of them use some of the geometric attributes computed automatically when a terrain model is present. These attributes can be viewed by choosing the *View Basin Geometrical Attributes* button. Lag time can be computed from one of several equations using these attributes, or by using a time computation coverage. These options are accessed from the *Compute Parameters – Basin Data* and *Compute Parameters – Map Data* buttons respectively (see section 15.3).

- CP - Peaking coefficient.

- TIME AREA CURVE - The time area curve defines the area of the watershed contributing runoff to the basin outlet as a function of time. This curve is defined by selecting the check box and then activating the *XY Series Editor* with the adjacent button. The time area curve can be computed automatically from a TIN (this method will not work for watersheds delineated from DEMs or Feature Objects) using the *Compute Time Area Curves* button.

10.9.3 SCS Dimensionless (UD)

Parameters for generating a unit hydrograph using the SCS dimensionless method include:

- TLAG - SCS lag time in hours. Several different equations have been published to determine the lag time of a basin. Many of them use some of the geometric attributes computed automatically when a TIN is present. Lag time can be computed from one of several equations using these attributes, or by using a time computation coverage. These options are accessed from the *Compute Parameters – Basin Data* and *Compute Parameters – Map Data* buttons respectively (see section 15.3).

10.9.4 Given Unit Hydrograph (UI)

A given unit hydrograph determined from a separate analysis can be input using the *XY Series Editor*. The given unit hydrograph must be derived for the same time interval as is specified on the IT record in the *Job Control* dialog.

10.9.5 Kinematic Wave (UK)

Distributed outflow from a basin may be obtained by utilizing combinations of three conceptual elements: overland flow planes, collector channels, and a main channel. These elements can be defined if the kinematic wave option is specified.

The first and second kinematic wave records can be used to distinguish between different properties such as pervious/impervious (grass/pavement). For each record, the following parameters can be supplied.

- L - Overland flow length.
- S - Representative slope.
- N - Manning's roughness coefficient.
- A - Percentage of sub basins area that this record represents (The total of the two records must sum to 100).

Losses - A loss method must be defined for each plane. Choosing the *Define Loss* button will present the *Standard Loss Method* dialog and allow a method to be chosen and parameters defined.

In addition to the kinematic wave records, collector channels and a main channel must be defined. Either kinematic wave (RK) or Muskingum-Cunge (RD) routing can be specified by selecting the appropriate radio button. A

dialog for defining the channels is accessed by choosing the *Define Channels* button. The main channel must be defined, whereas the two collector channels are optional. The following parameters are used for each channel:

- L - Channel length.
- S - Channel slope.
- N - Manning's roughness coefficient for the channel.
- CA - Contributing area to the channel.
- SHAPE - The characteristic shape of the channel.
- WD - Channel bottom width or diameter.
- Z - Side slopes if the channel type requires it.

For the main channel, only an eight point cross section as defined with the RC, RX, RY cards can be used.

A flag for routing upstream hydrographs can be specified for the main channel from within this dialog as well.

10.10 Snow Melt Data

When snow needs to be considered in the runoff analysis, snow melt data for a basin needs to be defined. *HEC-1* has two different methods for computing snowfall/melt simulations: the *Degree-Day* method, and the *Energy-Budget*.

To define data for a selected basin, choose the *Snow melt* Data button from the *Edit HEC-1 Parameters* dialog. The toggle at the top of the dialog (Figure 10-10) turns snow calculations on. For both methods the elevation or zone data, the coefficients, and temperature data must be defined.

The *Degree-Day* method is set up once these parameters have been defined. If the Energy-Budget method is toggled on then the Dewpoint, Shortwave Radiation, and Wind Speed data must be defined as well. losses should be defined when either method is used. These losses are used in conjunction with the LU or LE cards for normal basin losses. The check box at the bottom of the dialog allows losses to be turned on or off for a given simulation.

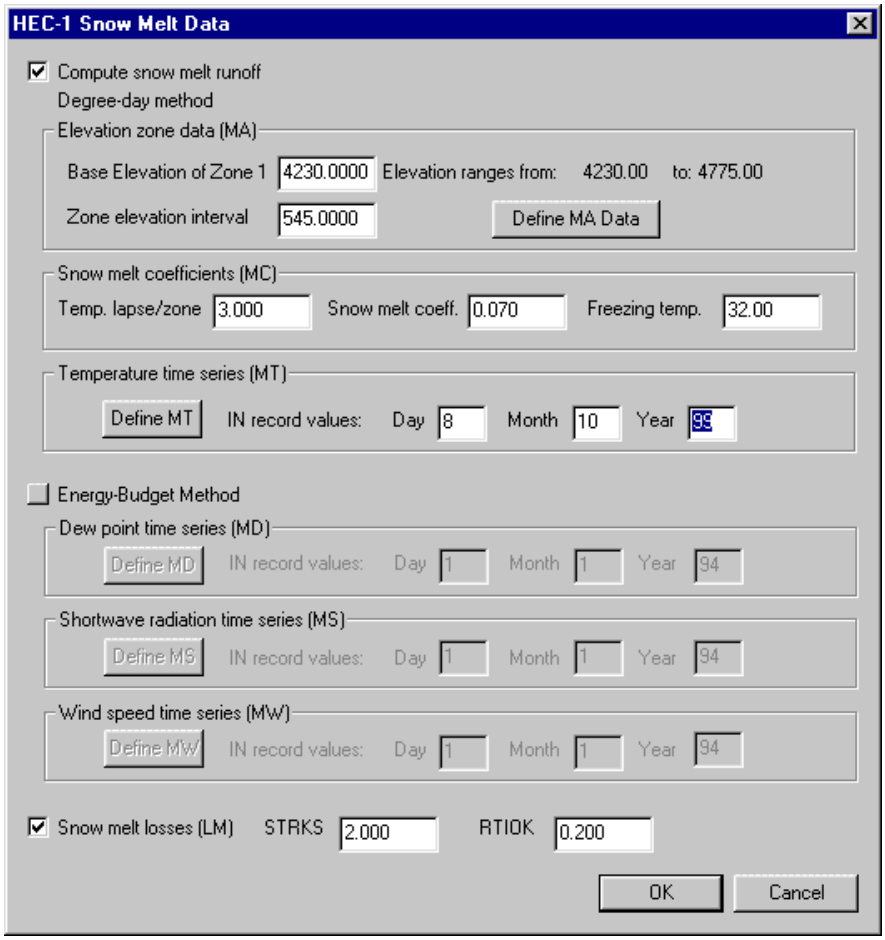


Figure 10-10 HEC-1 *Snow Melt Data Dialog.*

10.10.1 Elevation Zone Data (MA)

Snow computations are accomplished in *HEC-1* using separate, equally incremented, elevation zones within each basin. The number of elevation zones for which data must be defined is determined by specifying the base elevation of zone 1 and zone interval in the appropriate edit fields. The default values correspond to the lowest elevation and the range between the highest and lowest elevation (i.e. one elevation zone). More zones can be created by decreasing the interval, or lower elevations can be excluded from calculations by increasing the base elevation. Once the base elevation and interval are set, elevation zone data is defined by choosing the *Define MA Data* button. The number of zones which need to be defined is automatically determined and the appropriate edit fields are unhighlighted. The elevation zone parameters are as follows:

- AREA - The drainage area associated with this elevation zone.
- SNOPACK - The snow pack depth.

- AVEPRECIP - The normal annual precipitation in inches (mm) for this zone.

Areas for the elevation zones can be computed and supplied automatically using the compute areas button. The elevation fields are not part of the *HEC-1* input.

10.10.2 Temperature Time Series (MT)

The temperature time series is entered using the *XY Series Editor* where each value corresponds to the air temperature at the bottom of the lowest elevation zone for that interval. The starting date is determined from the IN record values in the edit fields corresponding to the temperature data. The starting time and time increment (also part of an IN record) are specified in the XY Series Editor using the *XY Options* dialog.

10.10.3 Dew Point (MD), Shortwave Radiation (MS), and Wind Speed (MW) Series

These three data records are only defined for the Energy-Budget method. Like the temperature time series, these three *HEC-1* records are defined using the XY Series Editor. Dates for IN records can be specified using the appropriate edit fields and the beginning time and time increment are defined using the *XY Options* dialog from within the XY Series Editor.

10.10.4 Losses (LM)

Losses are used in conjunction with the uniform (LU) or exponential (LE) loss methods. The parameter descriptions are as follows:

- STRKS - The starting value of the loss coefficient on the exponential recession curve for losses in in/hour (mm/hour) when used with the exponential loss rate (LE) or the uniform melt water loss rate (in/hour) when used with the uniform loss rate (LU).

- RTIOK - Rate of change of the loss-rate parameter computed as the ratio of STRKS to a value of STRKS after ten inches of accumulated loss when used with the exponential loss rate or not used when using the uniform loss rate.

10.11 Routing Data

Outlet points are used to define locations where hydrographs are combined and then routed downstream. The appropriate combined hydrograph (HC cards) stations are generated automatically when writing a *HEC-1* file. However,

routing data must be entered in order to simulate the movement of a flood wave through the river reaches or reservoirs. The effects of storage and flow resistance are accounted for in the shape and timing of the flood wave. In addition to these changes, volume may be lost due to channel infiltration. Most of the routing methods available in *HEC-1* are based on the continuity equation and some relationship between flow and storage or stage.

Routing data is entered by selecting an outlet and then selecting the Routing Data button from the *Edit HEC-1 Parameters* dialog. The different routing options available in this dialog are described in the following paragraphs.

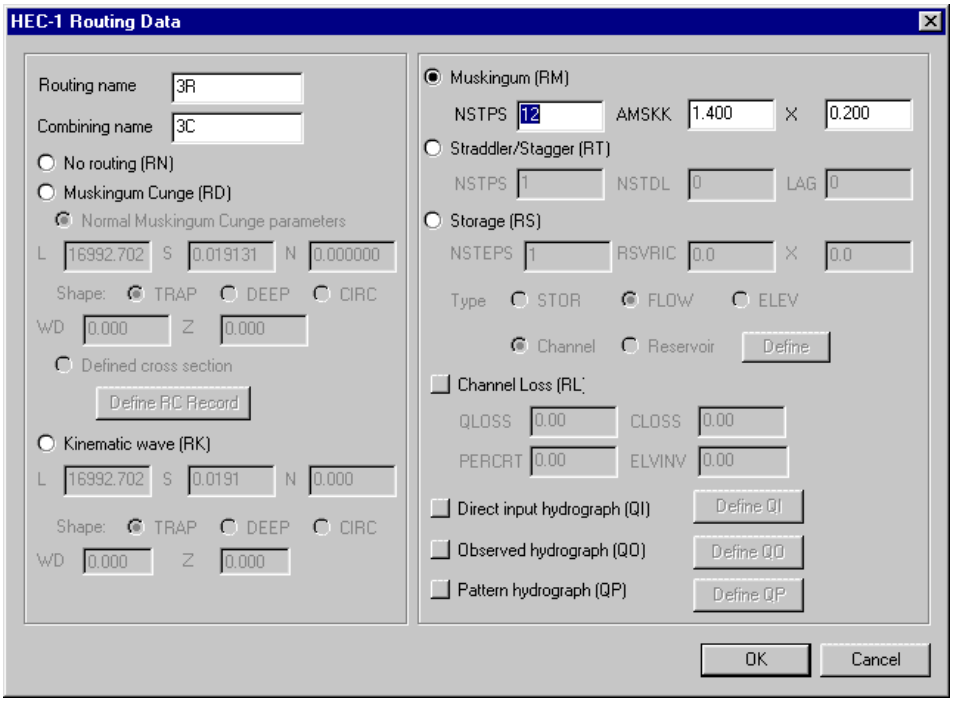


Figure 10-11 HEC-1 *Routing Data Dialog.*

10.11.1 Outlet Names (KK)

Since outlets are used for both types (combining and routing) of hydrograph stations in the *HEC-1* input file, a separate name for each type of hydrograph must be entered. The name should be six characters or less and correspond to the name used on the KK card to represent the appropriate hydrograph station.

10.11.2 No Routing (RN)

By default there is no routing at an outlet point. This allows for hydrographs to be combined without considering routing effects.

10.11.3 Muskingum (RM)

The Muskingum method is dependent primarily upon an input weighting factor. The parameters along with a short description of their meaning are as follows:

- NSTPS - The number of integer steps for the Muskingum routing.
- AMSKK - Muskingum K coefficient in hours for entire reach.
- X - Muskingum x coefficient.

Using the basin data computed by WMS when a TIN or DEM is used to delineate the watershed, the AMSKK and NSTPS coefficients can easily be estimated. AMSKK is essentially the travel time for the reach, which can be estimated by noting the length of the stream segment (displayed in the Muskingum Cunge dialog even though it is dimmed) and multiplying by an assumed channel velocity (1-5 ft/s would be appropriate for most natural channels). Of course you will need to convert the estimated travel times from seconds to hours before entering it into the AMSKK edit field. The NSTPS value is the number of time steps the flood wave is in the channel and can be determined by dividing AMSKK by the computational time step found in the *Job Control* dialog (again be sure that units are consistent).

10.11.4 Storage (RS)

Storage-discharge routing can be used to define either channel or reservoir routing. When this routing option is specified the appropriate data items are dimmed and additional radio buttons are used to determine whether channel or reservoir routing is to be used.

The following parameters must be defined regardless of the storage routing option specified.

- NSTPS - Number of steps to be used in the storage routing. Typically this is about equal to (reach length / average velocity) / time interval (NMIN). NSTPS is usually equal to 1 for a reservoir.
- ITYP - The next parameter RSVRIC can be entered in one of three different ways:
- STOR - Storage in acre-feet (1000 cu m).
- FLOW - Discharge in cfs (cms).
- ELEV - Elevation in feet (m).

- RSVRIC - Storage, discharge, or elevation as defined by ITYP corresponding to the desired starting condition at the beginning of the first time period as specified on the IT record in the Job Control dialog.

- X - Working R&D method not used.

Channel Routing

If the channel routing option is specified, the *Define* button will access a dialog shown which allows you to choose between normal depth and modified-Puls methods. If normal depth is specified, the following parameters must be defined for the RC record in the dialog shown in Figure 10-12.

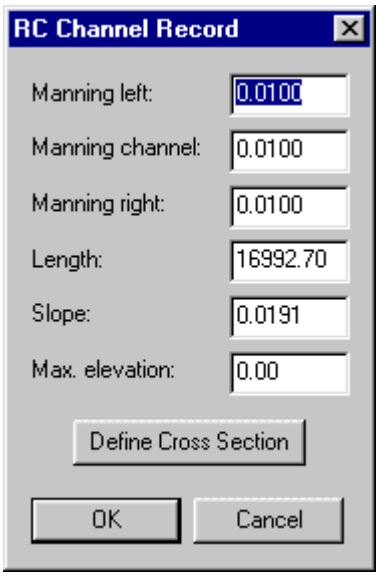


Figure 10-12 RC Channel Record Dialog.

- Manning's coefficients - Manning roughness coefficients for the channel and left and right overbanks.

- Length - The length of the river reach.

- Slope - The slope of the river reach.

- Max Elevation - The maximum elevation for which storage and outflow values are to be computed.

In addition to these parameters an eight point cross section must be defined using the *XY Series Editor*. The first two points define the left overbank, the third point defines the left bank, the fourth and fifth points define the channel itself, the sixth point defines the right bank, and the last two points define the right overbank.

If the modified-Puls method is chosen the volume (SV) outflow (SQ) relationship must be defined. Both records are defined using the *XY Series Editor*.

Reservoir Routing

If the reservoir routing option is specified then one method for volume and one method for outflow must be defined. The available methods are shown in Figure 10-13, and defined below.

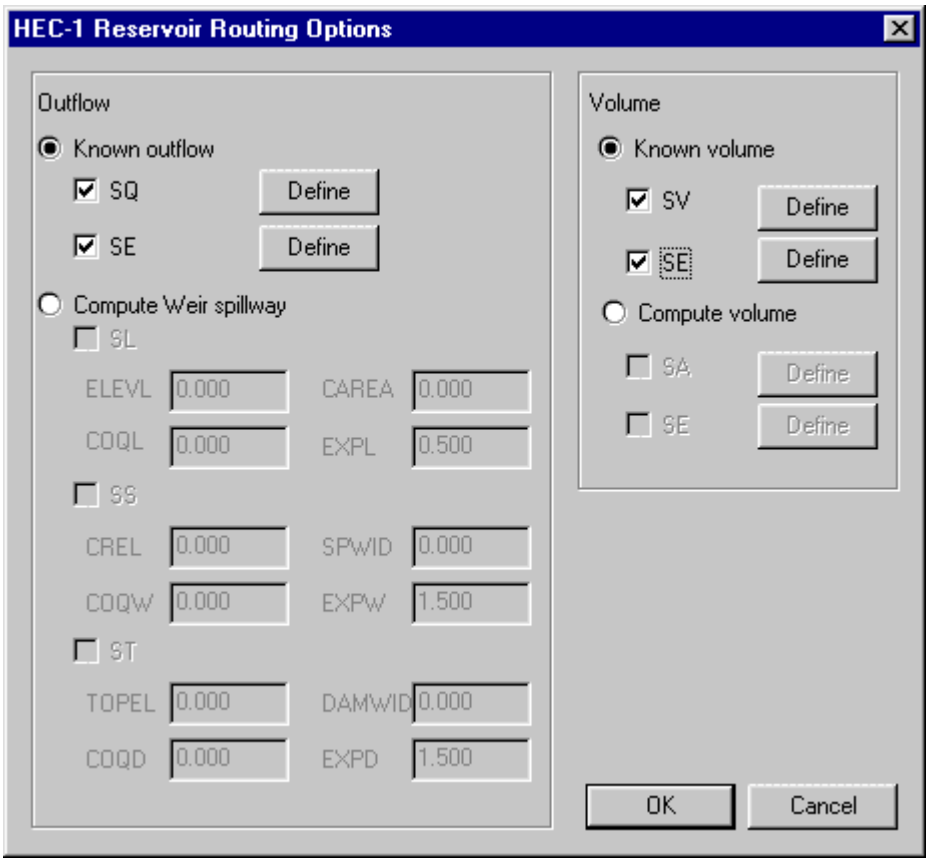


Figure 10-13 HEC-1 *Reservoir Routing Dialog.*

- Known volume - Define a known volume (SV) record using the *XY Series Editor*. Optionally, you can define the elevations (SE) which correspond to the known volumes.

- Computed volume - By defining an area (SA) elevation (SE) relationship the volume can be computed automatically by *HEC-1*. Both records are defined using the *XY Series Editor*.

If a TIN has been used to define the watershed a storage capacity curve can be generated and the information used to set up the volume/elevation (SV, SE) or area/elevation (SA, SE) data. See section 7.6.1 for more information.

The available methods for outflow include:

- Known outflow - Define a known outflow (SQ) record using the *XY Series Editor*.

- Computed Weir Spillway - A combination of data records are used to define spillway characteristics (SL, SS, ST). Parameter description for these different records are as follows:

Low-Level Outlet (SL)

- ELEVL - Center line elevation of downstream end of low-level outlet.

- CAREA - Cross-sectional area in square feet (square m) in the low-level outlet orifice equation.

- COQL - Discharge coefficient in orifice outlet equation.

- EXPL - Exponent of head in orifice equation.

Spillway Characteristics (SS)

- CREL - Spillway crest elevation. This value must be less than the highest elevation on the SE card for *HEC-1* to run properly.

- SPWID - Spillway length.

- COQW - Discharge coefficient in the spillway weir flow equation.

- EXPW - Exponent of head in the weir spillway flow equation, usually equals 1.5.

Dam Overtopping (ST)

- TOPEL - Elevation of the top of the dam at which overtopping begins.

- DAMWID - Length of the top-of-dam which is actively being overtopped.

- COQD - Discharge coefficient in the weir equation.

- EXPD - Exponent of head in the weir equation.

10.11.5 Straddler/Stagger (RT)

Parameters used to define the Straddler/Stagger or Tatum routing method are defined below.

- NSTPS - Should be one for Straddler/Stagger method or integer number of routing steps to be used for Tatum method.

- NSTDL - Integer number of intervals hydrograph is to be lagged in the Straddler/Stagger method or 0 if using Tatum method.

- LAG - Integer number of ordinates to be averaged in the Straddler/Stagger method or 2 if using Tatum method.

10.11.6 Kinematic Wave (RK) and Muskingum-Cunge (RD)

The Kinematic wave and Muskingum-Cunge are defined with essentially the same parameters. A brief description follows.

- L - Channel length.

- S - Channel slope.

- N - Manning's roughness.

- Shape - Characteristic channel shape.

- WD - Base width of the channel.

- Z - Side slope of channel.

If the Muskingum-Cunge method is selected, define the channel geometry using an eight point cross-section by specifying the appropriate radio button and selecting the Define RC Record button.

10.11.7 Channel Losses (RT)

Constant channel losses may be defined by defining values for the RL record. These parameters include the following:

- QLOSS - Constant channel loss in entire routing in cfs (cms). This value is subtracted from every ordinate of the inflow hydrograph.

- CLOSS - Ratio of remaining flow (after QLOSS) which is lost for entire routing. After subtracting QLOSS each inflow hydrograph ordinate is multiplied by (1-CLOSS).

- PERCRT - Percolation rate cfs/acre (cu m/sec-acre) for wetted surface area of channel. This option is used in conjunction with storage routing and requires SA or SV/SE records to be defined.

- ELVINV - Average invert elevation of channel L used to compute flow surface area for PERCRT.

10.11.8 Direct Input Hydrographs (QI)

Hydrographs can be input directly using the QI record, and then routed downstream using the different routing options. To do this, select the *Direct Input Hydrograph* option and define the QI record using the XY Series Editor.

10.11.9 Observed Hydrograph (QO)

This record is used to input an observed hydrograph for an optimization job (OR record). A QO hydrograph can be used in conjunction with a QI and QP hydrograph to optimize routing parameters. This hydrograph can be input by selecting the check box and then defining the hydrograph using the *XY Series Editor*.

10.11.10 Pattern Hydrograph (QP)

This option is used to input a pattern hydrograph for an optimization job (OR record). A QP hydrograph can be used in conjunction with a QI and QO hydrograph to optimize routing parameters. This hydrograph can be input by selecting the check box and then defining the hydrograph using the *XY Series Editor*.

10.12 Reservoirs

Reservoirs in *HEC-1* can be defined in few different ways, depending on the storage routing techniques that need to be modeled. The first tutorial on creating topologic trees outlines the different methods that can be used to represent reservoirs. The parameters required to define the reservoir are the same in all cases, and are described in the section 10.11.4 on reservoir routing above. The main difference is whether the reservoir stands alone by itself or whether the routing option of the outlet is used to define the reservoir.

10.13 Diversions

HEC-1 allows flow to be diverted from an outlet or drainage basin. This flow can be thought of as leaving the normal drainage system at that point. It can be retrieved at a downstream outlet where the diverted flow then contributes to the flow at that outlet. If no downstream retrieval outlet point is specified, the flow simply leaves the system at the diverted outlet point and never returns.

10.13.1 Editing Diversion Data (DT)

Diversions are created using a combination of the *Add* and *Retrieve Diversion* commands found in the *Tree* menu. Once created, data for the diversion can be defined and/or edited by selecting the *Diversion Data* button from the *Edit HEC-1 Parameters* dialog. This button will bring up the dialog shown in Figure 10-14.

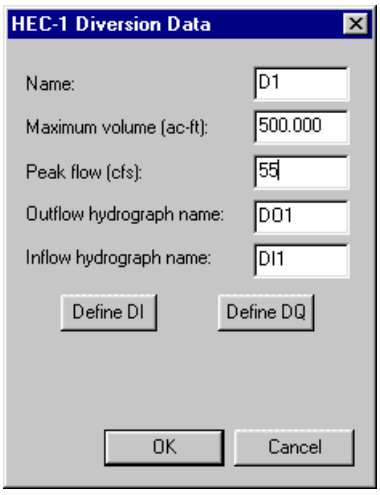


Figure 10-14 HEC-1 *Diversion Data Dialog.*

The following data should be defined for a diversion:

- **Name** - The name identification string of the diversion as used on the DT record. It is important to assign a unique name to each diversion in a given model because this name is used by *WMS* and by *HEC-1* to identify the diversion.

- **Max Volume** - Maximum volume of diverted flow in acre-feet (1000 cu m).

- **Peak Flow** - Peak flow that can be diverted in any computation period in cfs (cms).

- **Outflow name** - Name used on KK record where flow is diverted.

- **Inflow name** - Name used on KK record where flow is retrieved.

- The flow capacity of a stream flow diversion is specified using an **inflow** (DI) and **outflow** (DQ) **tables**. These tables are defined with the XY Series Editor by clicking on their respective define buttons.

10.14 Gages (PG)

The *Gages* command can be used to establish the position and rainfall accumulation for rainfall gages. Gages may be entered with or without a TIN present, but if a TIN is present then gage weights (using the Thiessen polygon method), for each basin are automatically computed when the *Compute Basin Data* command is chosen from the Drainage menu. When a TIN is not present the gage weights for each basin must be entered manually from within the *Precipitation* dialog.

In order for gage information to be used during the simulation, the basin precipitation type must be set to gage in the *Precipitation* dialog.

10.14.1 Defining/Editing Gages

The *Gage* dialog (Figure 10-15) consists of a list of defined gages as well as the fields necessary to define a new gage. When a gage is selected from the list, its values are copied into the appropriate fields for editing. When the *New* button is chosen, a new gage is created with default values. If the *Copy* button is chosen a new gage is created with the values of the currently selected gage. The *Delete* button can be used to remove the currently selected gage.

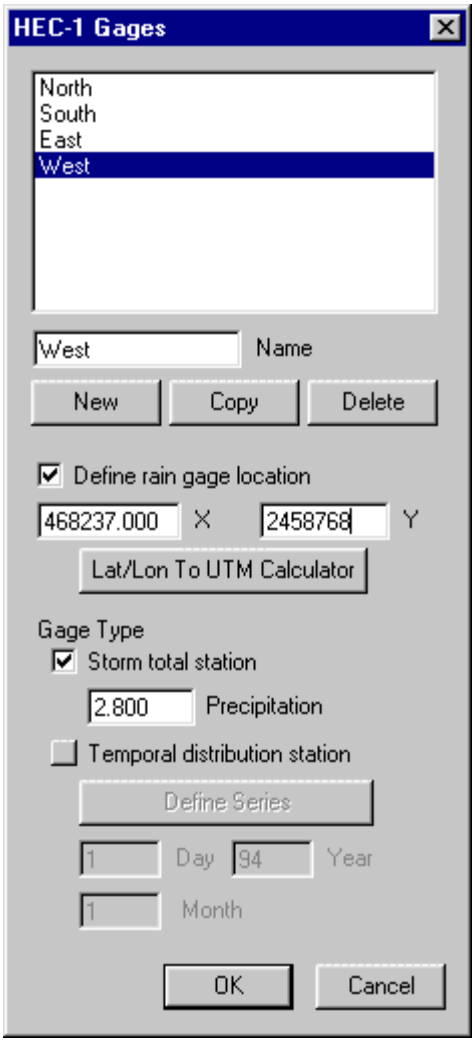


Figure 10-15 HEC-1 *Gage Dialog.*

10.14.2 The Gage Position

If a terrain model is being used then the X and Y position in a consistent coordinate frame needs to be entered in the appropriate edit fields. Once there are there are three or more defined gages, the Thiessen polygon network can be displayed (see the section titled *Drainage Display Options* in section 7.3). If a terrain model is not being used then the weights must be assigned manually and the position information is not important. In this case the *Define rain gage location* toggle should be turned off.

The rain gage coverage may also be used to define the position of the gages. Each feature point in the rain gage coverage is converted to a gage location and will automatically appear in the *HEC-1 Gages* dialog. See page 3-21 for more information on the rain gage coverage.

10.14.3 The Gage Type

A gage may be a storm total and/or temporal distribution (recording) station type. Recording stations allow for a continuous (incremental or cumulative) rainfall accumulation to be entered. The storm total station only allows for a single rainfall value for the event. A station may be both types if the distribution corresponds to the storm total value entered. However a more typical situation is to have several stations for which only a storm total is known and to which some type of standard distribution will be applied. To accomplish this, the storm total stations may be entered along with one "imaginary" gage that is used to define the distribution. When automatically computing weights, only storm total stations are used in the Thiessen network, and the distribution for each storm total station is found by locating the nearest distribution gage.

10.15 Reading And Writing *HEC-1* Files

Once a topologic tree has been created and all of the necessary data entered, an *HEC-1* input file can be generated by selecting the *Save HEC-1 File* command the File menu (*Save As* in the basic version). When writing the file, the proper order for computing, combining, and routing hydrographs is automatically determined. *HEC-1* can be run without any further editing of the input file generated by *WMS*. Because *WMS* does not allow input for all *HEC-1* options, it may be necessary to modify the file somewhat before execution. Hydrograph names defined on KK cards should not be changed, as they are needed to correctly read hydrographs generated by *HEC-1* back into *WMS* for post-processing.

WMS can read *HEC-1* input files so that data previously entered can be restored for basins and outlets. Names on the KK cards must match the basin or outlet names when reading the file for an existing terrain model.

Existing files generated outside of *WMS* can be read into *WMS* and a separate topological tree will automatically be generated for the watershed described in the file. Since *WMS* does not support all possible *HEC-1* card types, there may be some incomplete information. However, the basic structure of the watershed will be created and all possible data will be retained. Parameters from unrecognized cards, and/or hydrograph names, are ignored. More information on the limitations of reading existing HEC-1 simulations can be found in section 1.4.

10.16 Running An *HEC-1* Analysis

The version of *HEC-1* distributed with *WMS* can be run directly from *WMS* by using the *Run HEC-1* command in the *HEC-1* menu. Before running an *HEC-1* simulation you should run the model checker which will help you identify

serious and potential problems that should be corrected before a successful run of *HEC-1* can be made.

10.16.1 Model Check

The *Model Check* command should be issued once you feel that all necessary *HEC-1* data has been defined. It will report any possible errors/inconsistencies in your model so that corrections can be made prior to executing *HEC-1* (see Figure 10-16). Two types of information are provided as a result of this command. The first type is simply informational and provides things such as the starting time, time step, and total time of the simulation. You should verify that these parameters are what are intended. The second type of information messages are errors and must be corrected before an accurate *HEC-1* analysis can be performed. The list of checks made is not complete and just because no errors are reported does not insure that a successful and/or accurate analysis will be completed. We encourage you to report any additional checks that might be made as you work through various problems.

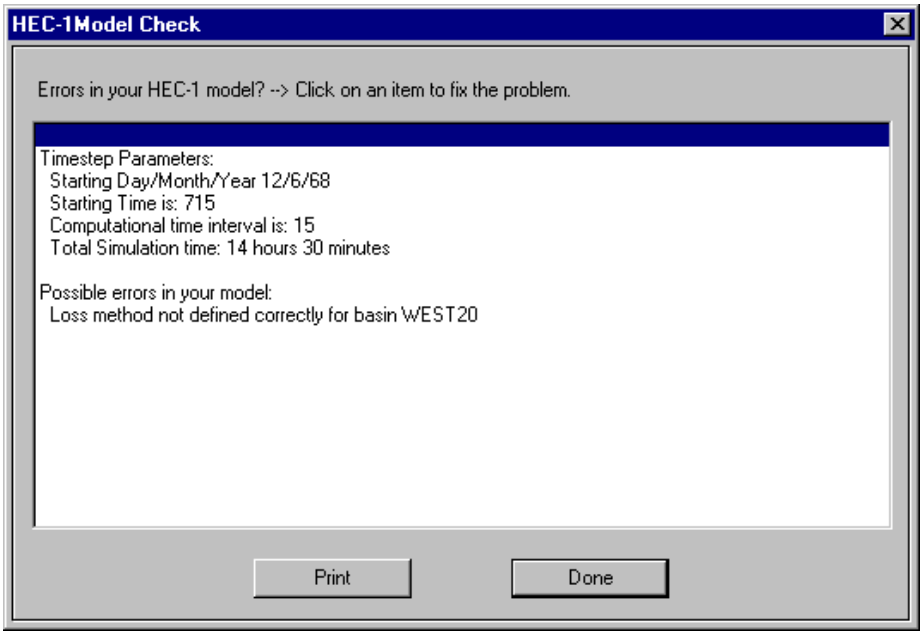

HEC-1Model Check
Errors in your HEC-1 model? --> Click on an item to fix the problem.
Timestep Parameters:
Starting Day/Month/Year 12/6/68
Starting Time is: 715
Computational time interval is: 15
Total Simulation time: 14 hours 30 minutes
Possible errors in your model:
Loss method not defined correctly for basin WEST20
Print
Done

Figure 10-16 *HEC-1* Model Check Dialog.

10.16.2 Run *HEC-1*

The *Run HEC-1* command will bring up a dialog (Figure 10-17) allowing you to specify three files which are necessary to run *HEC-1*. The first file is the *HEC-1* input file. The second is an ASCII output file generated by *HEC-1* and can be used to extract specific results values. It also contains important information which can be used to correct problems encountered when running *HEC-1*. The third file is a new name for the TAPE22 file, and will contain

hydrograph results for basins and outlets. You can view these results by reading this file with the *Open* command from the *Hydrographs* menu.

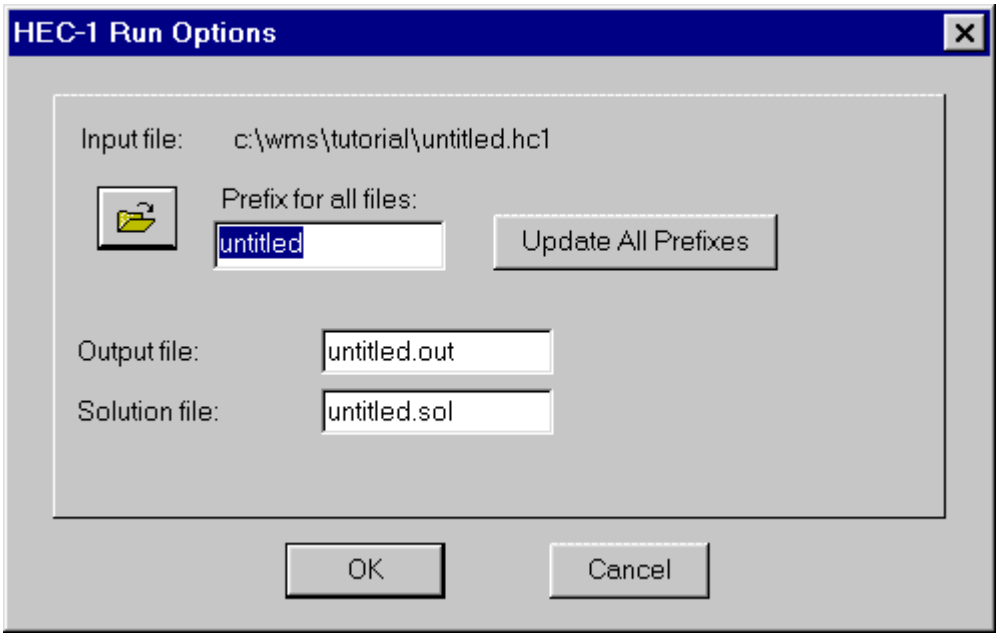


Figure 10-17 HEC-1 *Run Options Dialog.*

Once these files have been defined and you select OK, *HEC-1* will be executed. A separate window will appear and information about the *HEC-1* simulation will be reported. If you are running with Microsoft Windows you must close this window when *HEC-1* terminates. On UNIX workstation the window closes automatically.

If *HEC-1* is not executed successively when issuing this command then for PC computers be sure that the path to your HEC1.EXE file is included in AUTOEXEC.BAT, and for UNIX computers be sure the symbolic link to *HEC-1* as outlined in the Installation instructions has been appropriately defined.

If *HEC-1* does not run to a successful completion you can view the ASCII output file using the *View File* command in the *File* menu.

10.17 WMS and HMS

The Hydrologic Engineering Center (HEC) has replaced the HEC-1 analysis program with the HMS program. HMS includes most of the same capability as HEC-1, but has dropped some of the lesser used functions and added others. You can use WMS to delineate watershed data and define most parts of the HMS model. The *Export HMS Basin File* found in the *HEC-1* menu will

create the input file necessary to use data derived in *WMS* to perform modeling in HMS.

There are five different files that are (or can be) exported as part of the HMS simulation as shown in Figure 10-18.

1. The Project file is like the *WMS* super file in that it is used to keep track of the other files that make up the HMS simulation.

2. The Control file includes the job control data, or simulation global parameters such as time step, number of ordinates, id cards, etc.

3. The Basin file has all of the parameters for each hydrologic unit (basin, outlet, diversion, etc.).

4. The Precipitation file contains the information used to define the precipitation event for the simulation.

5. The Map file is a map or trace of the watershed and sub-basin boundaries. It is not a terrain model and therefore cannot be used to extract information such as area or runoff distance; it is only a "picture" of the watershed that is placed as a backdrop to the HMS schematic. If you have a watershed derived in *WMS* from a digital terrain model then you can export it as part of your HMS project. In order to export the map you must convert it to feature objects. If you create your watershed from feature objects or a DEM then it will already be in this format. If you create your watershed from a TIN then you will need to use the *Drainage Data -> Feature Objects* command found in the *Drainage* menu prior to exporting your HMS files.

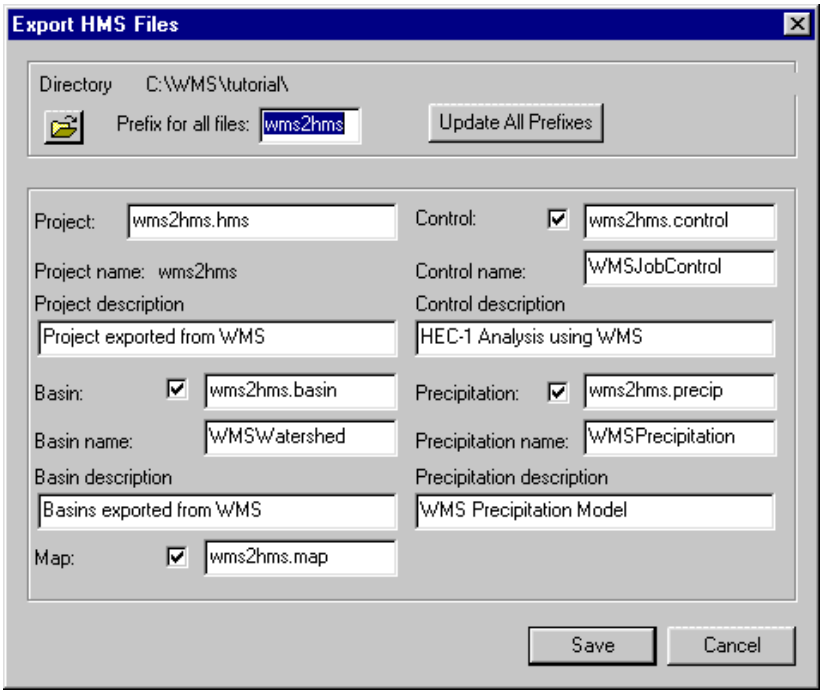


Figure 10-18 Export HMS Files Dialog.

CHAPTER 11

TR-20 *Interface*

11.1 Introduction

TR-20 models can be defined in *WMS* using the commands in the *TR-20* menu and resulting dialogs. There are six steps in defining a *TR-20* model using *WMS*:

11.1.1 Create a Topologic Tree

In the absence of terrain data, this is done using the commands to build the tree in the *Tree* menu. When using the TIN, Map, or DEM modules to delineate a watershed and sub-basin boundaries, a topologic tree is automatically created. Diversions can not be created directly on a TIN or in the Map module, but must be created using the commands in the *Tree* menu.

11.1.2 Define Job Control Parameters

Job control parameters are used to define the time and length of a simulation, output diagnostic controls, and other miscellaneous items. Definition of these parameters is discussed in detail later in this chapter.

11.1.3 Edit Basin/Outlet/Reservoir/Diversion Data

Parameters for all sub-basin outlets, reach routes, and diversions (hydrograph stations) are entered/edited using the *Edit TR-20 Parameters* dialog accessed from the *TR-20* menu. This dialog displays current values and lets you edit

values for the currently selected hydrograph station. More details are given later in this chapter.

11.1.4 Model Check

This step allows you to check your *TR-20* data prior to performing an analysis. It reports errors such as zero area, undefined precipitation, etc. These errors should be corrected by returning to previous steps before actually trying to run *TR-20*.

11.1.5 Run *TR-20*

WMS creates *TR-20* files compatible with any version of *TR-20*. However, a Windows (X-Windows on UNIX and MS-Windows on PC's) compiled version of *TR-20* is distributed with *WMS* so that it can be run without leaving the *WMS* environment.

The remainder of this chapter describes how data for creating a *TR-20* file is entered using *WMS*. For an in depth description of the *TR-20* runoff model itself, refer to the *TR-20* User's Manual.

11.1.6 View Hydrographs

Once a *TR-20* simulation has been run you can view resulting hydrographs using the commands in the *Hydrographs* menu. After viewing the hydrographs, you may wish to repeat the previous steps in order to calibrate a model, or look at different scenarios.

11.2 Tool Palette

The tools used for selecting outlets, basins, and diversions are defined in the TIN, DEM, Map, and Topological Tree modules. When entering *TR-20* data for basins or outlets, defined parameters will be assigned to all currently selected basins or outlets.

11.3 Job Control

The *Job Control* dialog box is used to define general information about the *TR-20* model. This dialog (Figure 11-1) is accessed by selecting the *Job Control* command from the *TR-20* menu. A list of the data entries for each item in the *Job Control* dialog with a short description for each follows.

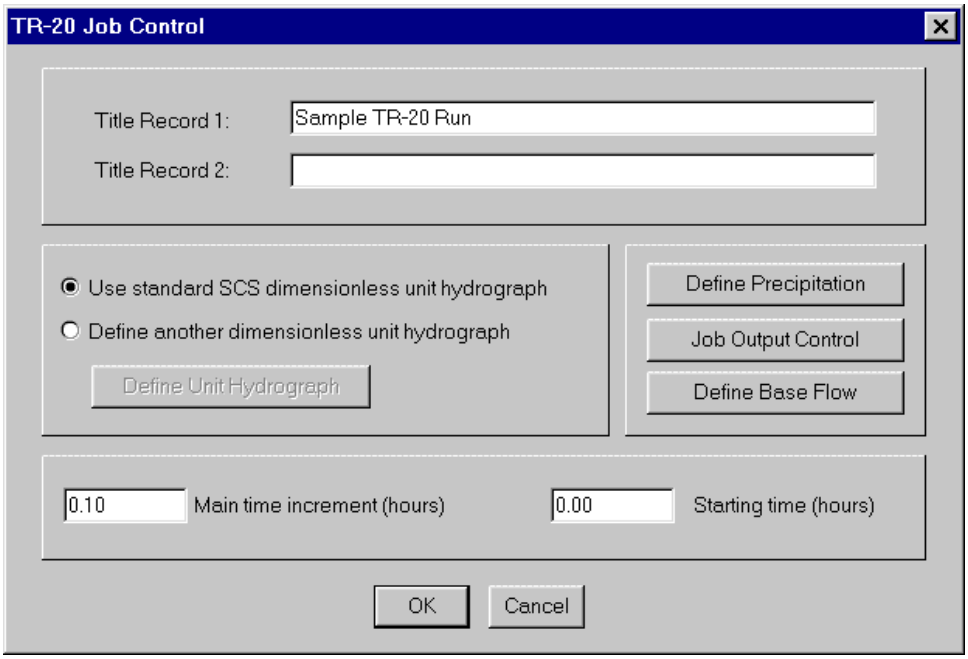


Figure 11-1 TR-20 Job Control Dialog.

11.3.1 Title

Enter a name and/or project description identifying the model. Two different title records up to 72 characters each can be entered. The title records will appear at the top of the *TR-20* input file.

11.3.2 Main Time Increment

The computational time increment defines the length in time between hydrograph ordinates. The interval should be specified in hours.

11.3.3 Starting Time

The starting time of the simulation is defined in this entry. The time should be specified as floating point number of hours past midnight. For example 7:45 am would be entered as 7.75 and 1:20 p.m. as 13.33.

11.3.4 Unit Hydrograph Definition

TR-20 offers two ways of defining the unit hydrograph used to compute runoff from your watershed. The first way is to use the standard SCS hydrograph provided with *TR-20*. Another way is to define your own input hydrograph. This can be done by selecting the *Define another dimensionless unit hydrograph* radio button and defining the unit hydrograph in the time series editor.

11.3.5 Precipitation

In *TR-20*, there are two different ways to define precipitation. Precipitation is defined by selecting the *Job Control* command from the *TR-20* menu. Then, select the *Define Precipitation* button to define precipitation over the entire watershed. You can define precipitation by using either a standard SCS rainfall distribution or by defining a custom rainfall distribution. If a custom rainfall distribution is used, you must define a custom rainfall distribution in the *XY Series Editor*. If one of the standard SCS rainfall distributions are used, you can select one of the following rainfall types:

- Emergency Spillway and Freeboard
- Type I: 24 hours
- Type II: 24 hours
- Type IA: 24 hours
- Type III: 24 hours
- Type II: 48 hours

For all the standard distributions, you must define the rainfall depth. For the dimensionless distribution, you must also define the rainfall duration. You only need to define the depth and/or duration of the rainfall for a custom distribution if one of the units entered in the *XY Series Editor* is dimensionless.

11.3.6 Base Flow

Base flow can be entered in *TR-20* from the *Job Control* dialog by selecting the *Define Base flow* button. There are two ways for defining base flow in *TR-20*. One way is to define a constant base flow. The second way is to define a triangular base flow in the watershed by specifying a volume in inches, a peak time in hours, and a base time in hours. By selecting the appropriate option, you can define either a triangular or a constant base flow for your watershed.

11.4 Entering/Editing *TR-20* Parameters

Attributes or parameters for all *TR-20* hydrograph stations are defined and/or later edited using the *Edit TR-20 Parameters* dialog. This dialog is accessed by selecting the *Edit TR-20 Parameters* command from the *TR-20* menu or when *TR-20* is the active model (see Figure 11-2) by double-clicking on basin, outlet, or diversion from the *Graphics Window*.

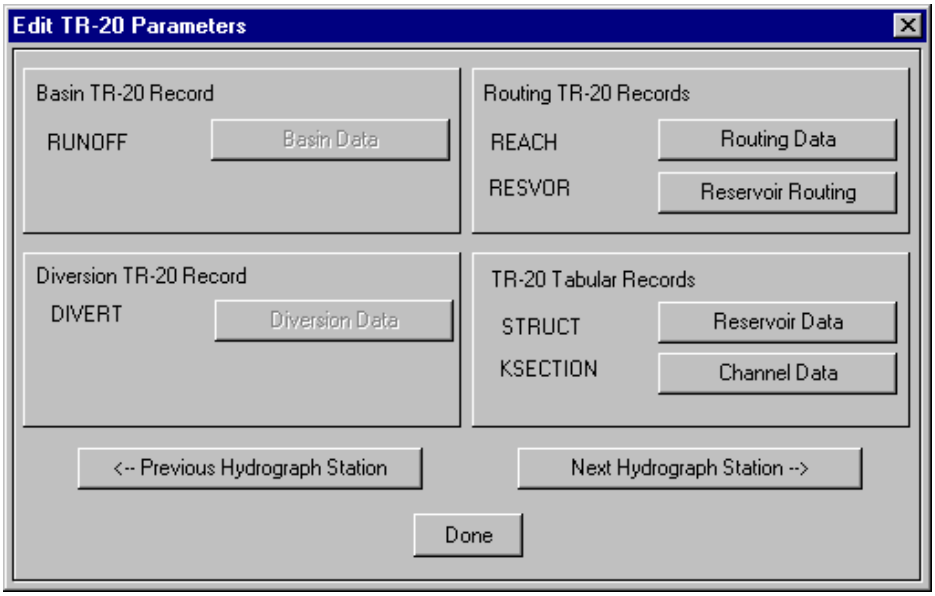


Figure 11-2 Edit TR-20 *Parameters Dialog.*

If a basin, outlet, or diversion is selected before issuing the command then data for that object is loaded for editing. This dialog lists the *TR-20* hydrograph station parameters which can be edited by selecting the corresponding button. When a hydrograph station is selected (basins/outlets/reservoirs/diversions) only the buttons which edit parameters associated with that hydrograph station are active, all others are dimmed.

Once the dialog appears it becomes part of the main screen until you select the Done button. Therefore, you can continue to select additional, or other hydrograph stations so that data for that object may be edited without exiting the dialog. You can use the Previous and Next Hydrograph Station buttons to cycle through hydrograph stations in the order they are computed by *TR-20.* While the dialog is up, all menu commands are active.

The remaining sections in this chapter cover the attributes which can be edited from each of the buttons in the *Edit TR-20 Parameters* Dialog.

11.5 Output Control

For each hydrograph station (basin hydrographs, combined hydrographs, routed hydrographs, reservoir hydrographs, and diversion hydrographs) different output controls can be specified. Selecting the *Output Control* dialog button from the individual hydrograph station dialog accesses this dialog. Entries that can be defined in this dialog are described below—

11.5.1 Peak Discharge and Runoff Volume

Selecting this option will output the following data at the selected hydrograph station(s):

- Peak discharge in cfs.
- Peak time in hours.
- Volume of water above constant base flow under the hydrograph in inches depth, acre-feet, and cfs-hours.
- Constant base flow value.

11.5.2 Discharge Hydrograph

Selecting this option will produce a discharge versus time hydrograph at the selected hydrograph station(s).

11.5.3 Elevation of Hydrograph

Selecting this option will produce an elevation versus time hydrograph in tabular form. This table will be produced for the hydrograph station(s) selected.

11.5.4 Flow Duration Data

Selecting this option will produce a table showing discharge versus duration of time that a discharge is equaled or exceeded. This table will be produced for the hydrograph station(s) selected.

11.5.5 Save Results for Summary Tables

This option puts the standard control operation results in summary tables 1 and 3 of the ECON2/URB1 generated files (see *TR-20* reference manual). Only the largest peak discharge for each hydrograph is listed in these files.

11.6 Basin Data (RUNOFF)

General information for each basin is entered by selecting a basin and choosing the *Basin Data* dialog. If multiple basins are selected, some fields will not be able to be edited (*Basin name* and *Area*), while others will be "grayed out". These other fields are in *Multi-select mode*…if you select and assign data to these fields, this data is assigned to all the selected sub-basins. A description of the data entry fields in this dialog is given below.

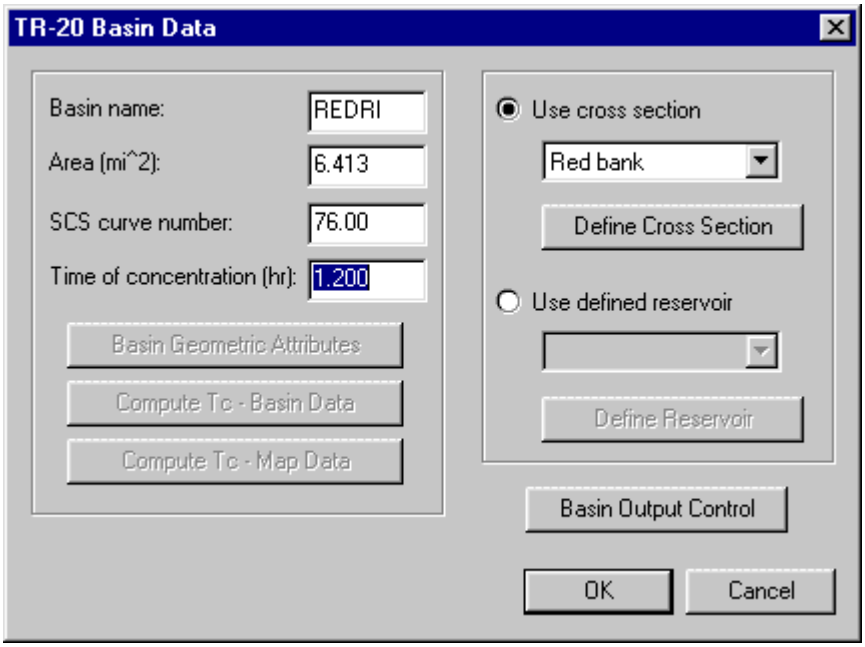


Figure 11-3 TR-20 *Basin Data Dialog*

11.6.1 Basin Name

Each hydrograph station should be identified with a unique name. This name is specific to *TR-20* files created by *WMS*, and is used to identify the basin in the file so that resulting hydrographs from a model run can be read back into *WMS* and associated with the basin. The name can not be more than six characters long. By default *WMS* uses the basin ID number followed by a "B" for the name, but a descriptive name is generally more useful.

11.6.2 Basin Area

When a TIN or Map-based model is present, basin areas and slopes can be computed automatically using the *Compute Basin Data* command from the Drainage menu. If a TIN or map data is not present, areas and slopes must be entered interactively using the topological tree as a map. Areas should be entered in square miles.

11.6.3 SCS Curve Number

The SCS curve number for rainfall/ losses on snow-free ground. **Note:** Composite Curve Numbers can be computed automatically, see section 15.2 for more information

11.6.4 Time of Concentration

Time of concentration in hours for the unit hydrograph. Several different equations exist for determining the time of concentration. Two different methods exist for computing the time of concentration…using computed basin data or using map data (see section 15.3). The list of basin geometric attributes computed automatically when basins have been delineated from a TIN can be useful in many of these equations. These attributes can be viewed and edited from within the *TR-20 Basin data* dialog by choosing the *Basin Geometrical Attributes* button.

11.6.5 Basin Geometric Attributes

When drainage data is computed, you can select this button to view and edit the geometric attributes of a basin, including the basin area, the flow length, and the basin slope. The different basin attributes can be viewed and edited only after computing the basin data. The basin data may be computed by selecting the *Compute Basin Data* command from the *Drainage* menu (*TIN* module), the *Compute Basin Data* command from the *Drainage* menu in the *DEM* module, or the *Update Basin Data* command from the *Feature Objects* menu in the *Map* module.

11.6.6 Compute TC – Basin Data

Pressing this button will bring up the *Unit hydrograph parameter computation dialog*. In this dialog, you can define the type of computation method to use in computing the time of concentration for *TR-20*. After exiting this dialog, the time of concentration, lag time, time to peak, and Snyder coefficient will be calculated. Many of these coefficients and times can be used in *HEC-1* models, and the time of concentration can be used in *TR-20* watershed models. By de-selecting the *Do not auto-recompute parameters* option, unit hydrograph parameters will be automatically re-calculated when the basin area is re-computed or other basin parameters are changed. See section 15.3.1 for more information about these equations.

11.6.7 Compute TC – Map Data

Times of concentration may also be computed using a time computation coverage. The basin time of concentration dialog is accessed by selecting the Compute TC – Map Data button. See section 15.3 for more information about computing the time of concentration with a time computation coverage.

11.6.8 Use Defined Cross Section

Select this option to use a defined cross section for your basin runoff computations. By selecting the *Define Cross Section* button, you can define a

cross section to use in the runoff computations. To define a cross section, the elevation, discharge, and end area must be defined for different intervals.

11.6.9 Use Defined Reservoir

Select this option to use a defined reservoir for your basin runoff computations. By selecting the *Define Reservoir* button, you can define a reservoir to use in the runoff computations. To define a reservoir, the elevation, discharge, and storage must be defined for different intervals.

11.7 Routing Data (REACH)

Outlet points are used to define locations where hydrographs are combined and then routed downstream. The appropriate combined hydrograph (ADDHYD records) stations are generated automatically when writing a *TR-20* file. However, routing data must be entered in order to simulate the movement of a flood wave through the river reaches or reservoirs. The effects of storage and flow resistance are accounted for in the shape and timing of the flood wave.

Routing data is entered by selecting an outlet and then selecting the Routing Data button from the *Edit TR-20 Parameters* dialog. Doing this will bring up the *TR-20 Routing Data* dialog (Figure 11-4). The different routing options available in this dialog are described in the following paragraphs.

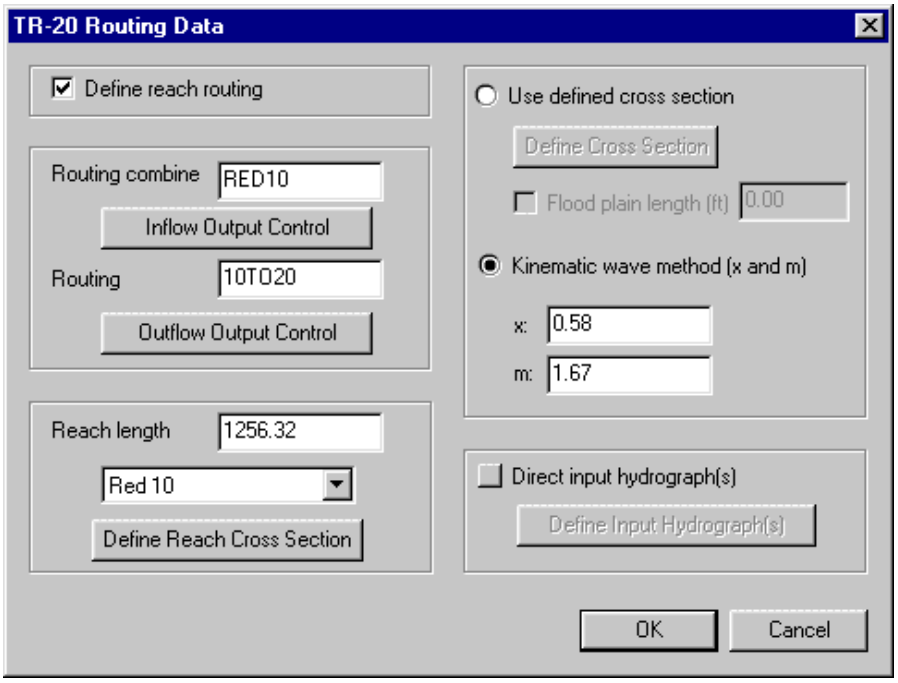


Figure 11-4 TR-20 *Routing Data Dialog*

11.7.1 Outlet Names

Since outlets are used for both types (adding and routing) of hydrograph stations in the *TR-20* input file, a separate name for each type of hydrograph must be entered. The name should be eight characters or less and is used to read hydrographs from the results file.

11.7.2 Reach Length

This value should be the length, in feet, along the river reach for which routing takes place. If a TIN, DEM, or Map-based watershed model is present this value is automatically computed and assigned to this field whenever the *Compute Basin Data* command is executed. A cross section must be assigned to the reach length assigned. The cross section editor is discussed in section 11.7.3.

11.7.3 Routing Method

TR-20 has two different routing methods to choose from. If cross-sectional data is available it can be used to establish routing parameters using a "m-value" method. If the cross-section data are not available a Kinematic wave method may be used instead. The method for each outlet is determined by the radio group selection.

Using Defined Cross Sections

With this routing method a typical cross-section for each reach (outlet) must be defined. This is done using the *TR-20 Cross Section Data* dialog, and is accessed by selecting the *Define Cross Section Data* button. This dialog is shown in Figure 11-5 and a description of the different options follows.

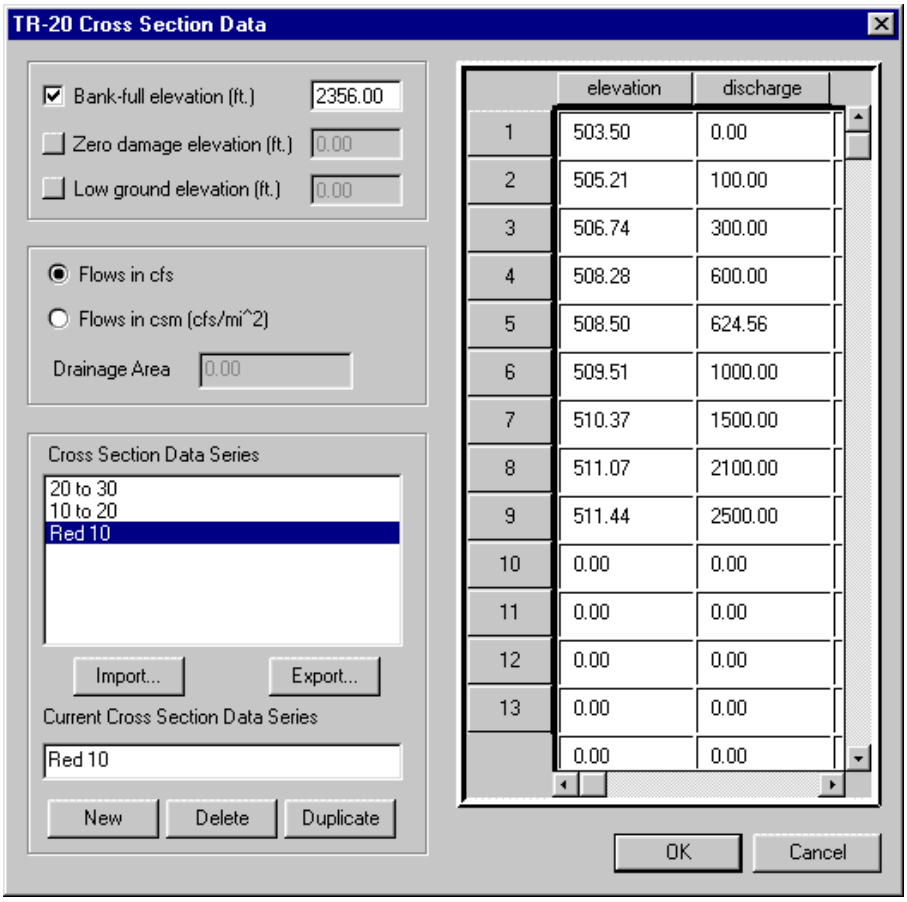


Figure 11-5 TR-20 *Cross Section Data Dialog.*

Bankfull Elevation

If you enter the bank full elevation here, it will trigger a warning message in the *TR-20* output file if less than two cross section data points are below bank full. Entering the bankfull elevation is optional.

Zero Damage Elevation

This is used with the *TR-20* flow duration analysis to flag results at this elevation. Entering the zero damage elevation is optional.

Low Ground Elevation

The low ground elevation is the lowest flood plain elevation in a cross-section. The low ground elevation, which is optional, must not be higher than the bankfull elevation.

Flow Units

The flow can be entered in cubic feet per second per square mile (csm) or cubic feet per second (cfs). To select one of these flows, simply select the type of flow you want by selecting the appropriate radio button in the *Define*

Cross Section dialog. If flow is defined in csm, you must define the drainage area of the basin in square miles.

Defining Cross Sections

Once a cross section has been defined in TR-20, this cross section can be used in other locations in the TR-20 model. A cross-section defined for an earlier outlet can be used later by another outlet simply by selecting the name from the text window or the drop-down box in the *TR-20 Routing Data* dialog.

Kinematic Wave Method

The kinematic wave method uses two different rating coefficients, x and m, in the following equation:

$$Q = xA^m \quad \text{..... 11.1}$$

where Q is the discharge, A is the valley storage area divided by length and x and m are the coefficient and exponent of the relationship describing the reach and maximum inflow hydrograph peak discharge.

11.7.4 Output Control

Control of the output files can be specified individually for each of the reaches or outlets. The output control parameters for the inflow hydrograph corresponds to the ADDHYD record whereas the parameters for the outflow hydrograph correspond to the REACH record. Output control parameters for all hydrograph stations are identical and are discussed in an earlier section.

11.7.5 Direct Input Hydrographs

Hydrographs can be input directly, and then routed down stream, using the different routing options. To do this, select the *Direct Input Hydrograph* option. Hit the *New* button in the *Define input hydrographs* dialog to define a new hydrograph. The base flow and contributing drainage area for the hydrograph should be entered for computation purposes.

The hydrograph may be edited by selecting the *Edit* button after selecting the hydrograph to edit. For each input hydrograph in *TR-20*, the discharge in cfs at each time interval must be entered.

11.8 Reservoirs

TR-20 allows you to route a hydrograph through a reservoir using the *RESVOR TR-20* file card. You can define these reservoir routing parameters in the *TR-20 Reservoir Routing* dialog (Figure 11-6). Reservoirs and other types of

structures (such as detention basins) can be defined from the *TR-20 Reservoir Data* dialog (Figure 11-7). The *TR-20 Reservoir Routing* dialog is explained below—

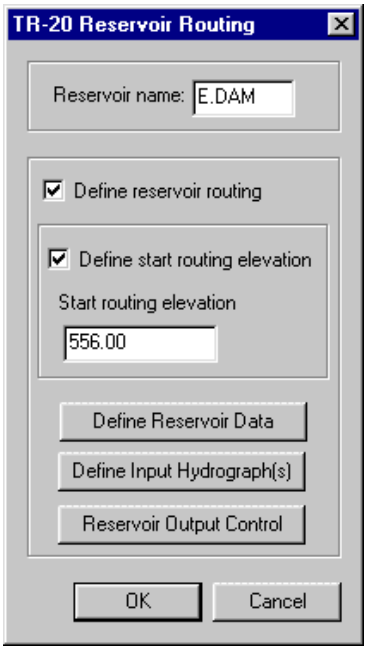


Figure 11-6 TR-20 *Reservoir Routing Dialog*

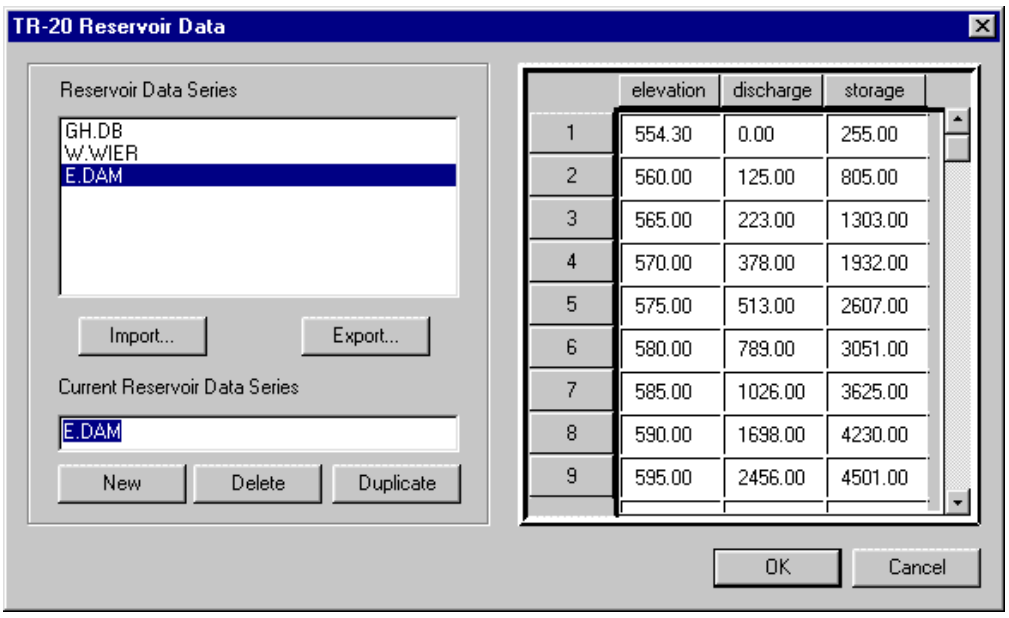


Figure 11-7 TR-20 *Reservoir/Structure Data Dialog*

11.8.1 Name

The reservoir name is used to associate resulting hydrographs with the appropriate reservoir when reading a hydrograph file after a *TR-20* run. The name should be unique and no longer than eight characters.

11.8.2 Defining Routing

Routing can be toggled on by selecting the *Define Reservoir Routing* button. After selecting the *Define Reservoir Routing* button, reservoir routing will be defined at the selected tree nodes. If this option is not selected, routing will not be defined at the selected tree node.

11.8.3 Start Routing Elevation

The start routing elevation is the water surface elevation, in feet, that routing begins for the reservoir or structure.

11.8.4 Defining Reservoir Data

You can define reservoir data by pressing the *Define reservoir data* button. This will bring up a dialog where you can define the elevation-discharge-storage relationship for the reservoir (see Figure 11-7). You can enter up to 20 elevation-discharge-storage relationships for each reservoir. To assign a particular set of data to a reservoir, select the reservoir you want and select the *OK* button on the *TR-20 Reservoir data* dialog.

11.8.5 Input Hydrograph

One or more input hydrographs can be defined for a reservoir. *WMS* will combine any input hydrographs with other input hydrographs to the reservoir. The resulting hydrograph will then be routed through the reservoir. This option is useful for defining measured stream flows into a reservoir.

11.8.6 Output Control

Control of the output files can be specified individually for each of the reservoirs. Output control parameters for all hydrograph stations are identical and are discussed in an earlier section.

11.9 Diversions

TR-20 allows flow to be diverted from an outlet or drainage basin. This flow can be thought of as leaving the normal drainage system at that point. It can

be retrieved at a downstream outlet where the diverted flow then contributes to the flow at that outlet. If no downstream retrieval outlet point is specified, the flow simply leaves the system at the diverted outlet point and never returns. The diversion data is defined in the *TR-20 Diversion Data* dialog (Figure 11-8).

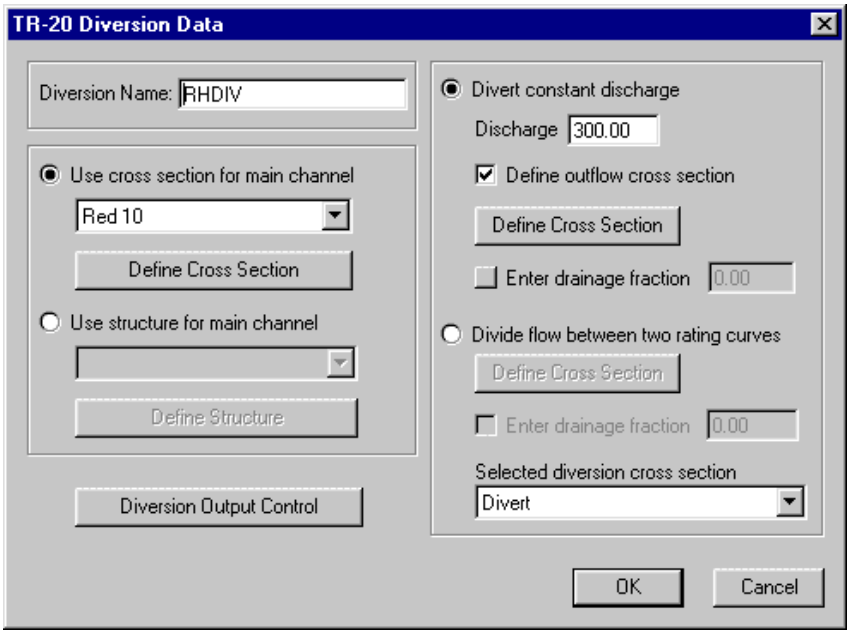


Figure 11-8 TR-20 *Diversion Data Dialog*

11.9.1 Name

The diversion name is used to associate resulting hydrographs with the appropriate diversion when reading a hydrograph file after a *TR-20* run. The name should be unique and no longer than six characters.

11.9.2 Diversion Inflow Parameters

The inflow for the diversion can be determined in one of two ways: either by using a cross-section for the main channel or by using a structure for the main channel. Cross-sections are defined using the same dialog as for reaches as described in the earlier section on channel routing. Structures are defined in the same way as structures for reservoir routing. The total amount of flow diverted can be defined in one of two ways.

11.9.3 Diversion Outflow Parameters

The outflow for a diversion can be specified in one of two ways. In the first method, a constant flow is diverted. For this method, you must define what

this constant outflow is. You can also define a cross section and the decimal fraction of the drainage area to be associated with the main channel output hydrograph (the drainage area fraction) here. In the second method of defining a diversion, you define the cross section for the diversion. *TR-20* uses a rating curve to divide flow between the two cross sections that comprise the diversion. You can define the decimal fraction of the drainage area to be associated with the main channel output hydrograph (the drainage area fraction) using this method as well.

11.9.4 Output Control

Control of the output files can be specified individually for each of the diversions. Output control parameters for all hydrograph stations are identical and are discussed in an earlier section.

11.10 Reading And Writing *TR-20* Files

Once a tree has been created and all of the necessary data entered, a *TR-20* input file can be generated by selecting the *Save TR-20 File* command the *TR-20* menu. When writing the file, the proper order for computing, adding, and routing hydrographs is automatically determined. *TR-20* can be run without any further editing of the input file generated by *WMS*. Because *WMS* does not allow input for all *TR-20* options, it may be necessary to modify the file somewhat before execution. Hydrograph names must be defined in columns 73-80 of all RUNOFF, REACH, DIVERT, ADDHYD, and RESVOR, records and GRAPHICS must be defined in columns 61-68 of the JOB record before output of *TR-20* can be read back into *WMS* for display in the *Hydrograph Window*.

Existing files generated outside of *WMS* can be read it into *WMS* and a separate topological tree will automatically be generated for the watershed described in the file. Since *WMS* does not yet support all possible *TR-20* card types, there may be some incomplete information. However, the basic structure of the watershed will be created and all possible data will be retained. If multiple computations or runs of *TR-20* are specified in a single file, WMS will prompt you for which one you want to use. With *WMS* only a single computation sequence may be defined. If additional computations are desired, the parameters can easily be changed within *WMS* and another run of *TR-20* made.

11.11 Running A *TR-20* Analysis

The version of *TR-20* distributed with *WMS* can be run directly from *WMS* by using the *Run TR-20* command in the *TR-20* menu. Before running a *TR-20* simulation, you should run the model checker. The model checker will help

you identify serious and potential problems. These problems should be corrected before a successful run of *TR-20* can be made.

11.11.1 Model Check

The *Model Check* command should be issued once you feel that all necessary *TR-20* data has been defined. It will report any possible errors/inconsistencies in your model so that corrections can be made prior to executing *TR-20*. The list of checks made is not complete and just because no errors are reported does not ensure that a successful and/or accurate analysis will be completed. We encourage you to report any additional checks that might be made as you work through various problems.

11.11.2 Run *TR-20*

The *Run TR-20* command will bring up a dialog allowing you to specify 3 files which are necessary to run *TR-20*. The first file is the *TR-20* input file. The second is an ASCII output file generated by *TR-20*. This output file can be used to extract specific results. It also contains important information which can be used to correct problems encountered when running *TR-20*. The third file will contain hydrograph results for basins and outlets. You can view these results by reading this file with the *Open* command from the *Hydrographs* menu.

Once these files have been defined and you select OK, *TR-20* will be executed. A separate window will appear and information about the *TR-20* simulation will be reported. If you are running with Microsoft Windows you must close this window when *TR-20* terminates. On UNIX workstations the window closes automatically.

If *TR-20* is not executed successfully when issuing this command then be sure that the path to your *TR20.EXE* file (*tr20* for UNIX) is located in the same directory as your WMS executable file.

If *TR-20* does not run to a successful completion you can view the ASCII output file using the *View File* command in the *File* menu.

CHAPTER 12

TR-55 *Interface*

12.1 Introduction

TR-55 models can be defined in *WMS* using the commands in the *TR-55* menu and resulting dialogs. The *TR-55* model has long been used to determine the increase in runoff resulting from the development of rural land into urban land. The *WMS* interface to *TR-55*, combined with the basin delineation and time of concentration calculations from feature objects (see section 15.3), makes it simple to set up and run several different scenarios in a relatively short time period.

This chapter describes the mechanics of the *WMS* interface to *TR-55* and is not a complete reference to the methodology, uses, and limitations. For more in depth information you should read "Technical Release 55: Urban Hydrology for Small Watersheds, 2nd Edition."

12.2 Input Requirements for *TR-55*

To run a *TR-55* simulation you select the *Run Simulation* options from the *TR-55* menu. The *TR-55* dialog (see Figure 12-1) is a floating dialog. This means that all tools and menu commands are available while this dialog is active. The fields that are active depend on whether a basin, an outlet, or nothing is selected. You must define the data for each basin, or outlet by selecting it and then entering the appropriate values in the corresponding edit fields. Some of the values (area, CN) may be computed/mapped automatically by *WMS* if you are using a watershed model delineated by *WMS*.

12.2.1 Initializing a *TR-55* Simulation

The first time you open the *TR-55* simulation dialog (no matter whether you have a basin, an outlet, or nothing selected) the only option available in the *TR-55* dialog is the *Initialize TR-55 Data* button. In order to allocate memory in *WMS* for *TR-55* variables you must initialize the data once. Should you decide to work with a different model you may want to choose the *Delete TR-55 Data* button. If you are planning on using a TIN, DEM, or feature objects to delineate and compute watershed parameters you should initialize *TR-55* data first so that computed parameters can be stored in the *TR-55* data structures.

12.2.2 Basin Data

The following information must be determined and appropriate values entered for each sub-basin.

1. Drainage Basin Area.
2. Time of Concentration.
3. SCS (NRCS) Curve Number.
4. 24-hour rainfall depth and an appropriate time distribution curve.

Optionally the effects of water bodies can be determined by entering the percent of the watershed covered by lakes/ponds.

12.2.3 Outlet Data

WMS allows you to define your watershed as a single basin or you may also subdivide the watershed into multiple sub-basins. Besides computing peak flows using the standard *TR-55* equations, storm hydrographs may also be computed for each basin and then combined/lagged to downstream outlets (junctions) using the *TR-55* tabular hydrograph method.

When you create a watershed with multiple sub-basins you must define the time of travel from one outlet (junction) to the next. The travel time for an outlet is defined by selecting the outlet and then entering the appropriate travel time. You may wish to use a time computation arc to compute the travel time (see section 15.3).

12.3 Using the *WMS* Interface to Run *TR-55* Simulations

Once all of the input necessary to run a *TR-55* simulation is defined, peak flow and hydrograph computations are made using the dialog shown in Figure 12-1.

The first time you select the *TR-55 data window* lists the remaining information that must be entered before a peak flow is calculated. Once all of the items listed in this window are entered, *WMS* performs the calculation for peak discharge and displays the result in the same data window. Additional information (help) about any line selected in the *TR-55 data window* is displayed in the *TR-55* help window. In Figure 12-1 the help shows the equation for the peak discharge in *TR-55* (selected line in the data window).

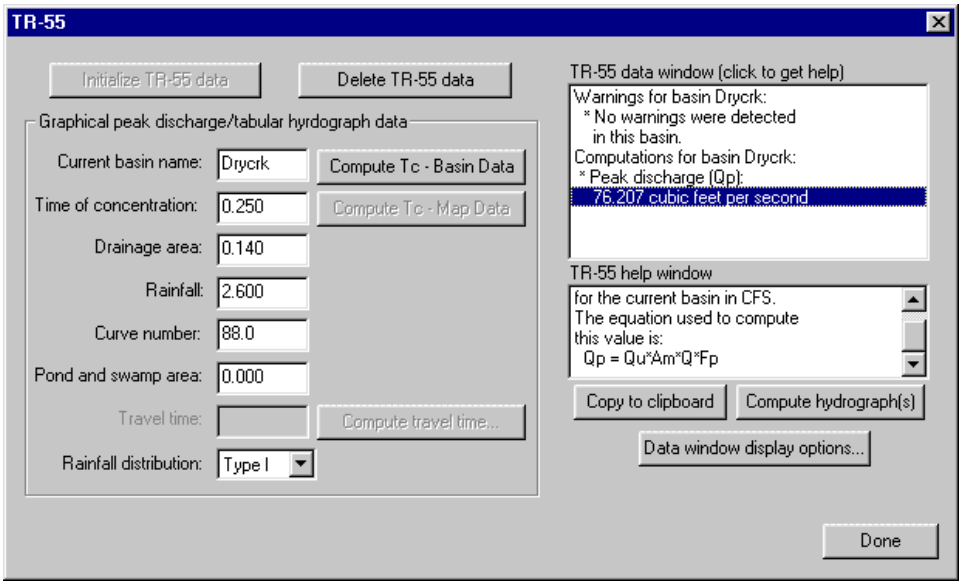


Figure 12-1 TR-55 *Dialog.*

The *Data window display options…* button allows you to control which information are displayed in the data window when the peak discharge is computed. The primary purpose for including the different display options (shown in Figure 12-2) is so that all results can be copied to the clipboard and then pasted into a report document.

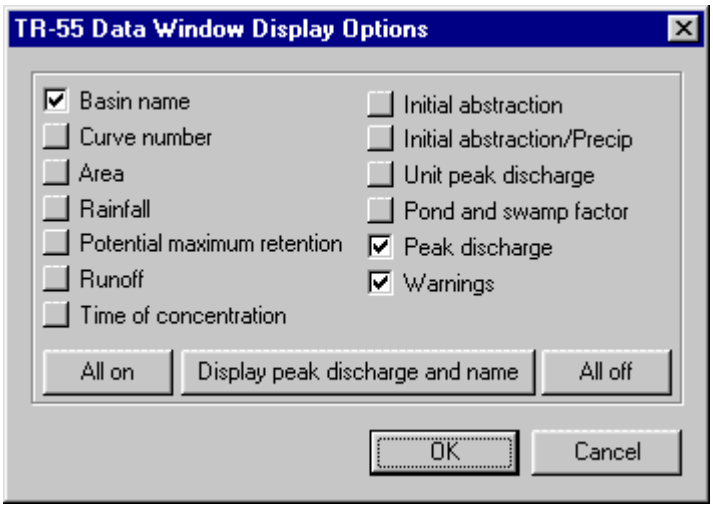


Figure 12-2 TR-55 *Data Window Display Options Dialog.*

The input fields required to run a *TR-55* simulation are defined in the next several sections.

12.3.1 Basin Name

The basin name is used to identify individual basins within a larger watershed. As with all basins created automatically in *WMS*, the name is defaulted to "ID"B where ID is an internal identification number and B stands for basin. The name can be changed to something more identifiable at any time. While *TR-55* does not require that basin names are unique, other models supported by *WMS* do.

12.3.2 Time of Concentration

Three different options exist for determining the basin time of concentration (tc).

1. The time of concentration can be computed "outside" of *WMS* and entered into the appropriate edit field.

2. Second the *Compute Tc – Basin Data* button can be chosen and one of the time of concentration (or lag time) equations specified (this option is only available when you have computed basin data from either a TIN or a DEM). The available equations in *WMS* are described in section 7.7.11.

3. Finally a series of time computation arcs may be used to define overland, sheet, and channel flow with in a basin and then travel times for each arc are summed to compute the total travel time or time of concentration for the basin. The *Compute Tc – Map Data* button accesses the dialog which allows you to combine arcs within the currently selected basin to compute a time of concentration. A description of the time computation coverage can be found in section 3.2.6 and it's use is described in more detail in section 15.3.

12.3.3 Drainage Area

Enter the area of the basin in this edit field. If you have delineated a watershed from feature objects, a DEM, or a TIN, the drainage area can be computed and will automatically be updated in this field.

12.3.4 Curve Number

The NRCS (SCS) Curve Number (CN) should be entered in this field. CN is a function of hydrologic soil group and land use. Composite CN's may be

computed for each basin and mapped to this field. For more information on computing CN's from GIS data layers see section 15.2.

12.3.5 Rainfall

Both the total rainfall and a dimensionless mass distribution must be specified to define precipitation for a basin. The total rainfall is entered in the rainfall edit field, and the distribution type is specified by selecting one of the standard NRCS (SCS) rainfall distribution types from the drop down list. The *TR-55* reference manual shows a map (Appendix B-2 of the *TR-55* reference) of distribution types for the US.

12.3.6 Pond/Swamp Factor

The pond and swamp factor will affect the peak discharge. The percentage of area for the selected basin that is made up of ponds, lakes, swamps, etc. should be entered in the appropriate edit field.

12.3.7 Travel Time

The travel time is the amount of time a flood wave takes to move from the outlet to the downstream most point of the watershed. This is different than other models in *WMS* which has you enter the travel time from outlet to outlet. Computed hydrographs at each basin and each outlet point represent the amount of the hydrograph at the outlet point contributed from each basin or each outlet point. Thus, the hydrographs you see when you run *TR-55* in *WMS* are already lagged. The upstream hydrographs are then simply added together to determine the hydrograph at the watershed outlet point. No lagging is done in *WMS*; tabulated *TR-55* hydrographs are pre-lagged.

Like time of concentration, travel times between outlets may be computed using a series of feature objects with equations such as Manning's defined for each (see section 15.3).

12.4 Computing Hydrographs

Using the *TR-55* tabular hydrograph method (see *TR-55* reference manual for details), hydrographs for a selected basin or outlet may be computed. The *Compute Hydrograph(s)* button is chosen once all of the necessary input for basins and outlets have been entered and a new set of hydrographs will be computed.

If you have a basin selected when you compute hydrographs then a hydrograph for only the selected basin will be computed. However, if you have an outlet selected when you compute hydrographs then a new hydrograph for each

upstream basin will be computed along with the routed hydrographs at outlet points.

You may notice a slight difference between hydrographs computed by *WMS-TR-55* implementation and those in the *TR-55* reference manual. This difference occurs because the Ia/P value is rounded to the nearest hydrograph value for the standard version of *TR-55*. In *WMS* hydrographs are determined by linear interpolation between two hydrographs using the actual Ia/P value.

12.5 Copying Results to the Clipboard

The *Copy to Clipboard* button can be used to copy the information from the *TR-55* data window to the clipboard for pasting into a word processing document. The data window display options (see Figure 12-2) determine which things are visible and ultimately copied to the clipboard.

CHAPTER 13

Rational Method

13.1 Introduction

WMS includes an interface to the rational method which can be used for computing peak flows on small urban and rural watersheds. The interface includes the capability to combine runoff from multiple basins. Two different methods for determining peak flows/hydrographs at downstream confluences are available.

Traditionally a time of concentration is determined at a downstream confluence by determining the longest combination of time of concentration and routing travel time. Given a time of concentration for the outlet, a rainfall intensity can be determined from a rainfall-intensity-duration curve and a peak flow computed. The hydrograph for the confluence is then determined in the same manner they are determined for sub-basins; by using the peak flow, time of concentration, and a dimensionless hydrograph (see section 13.3.4).

Alternatively, hydrographs for the sub-basins can be computed and then routed (lagged) and combined by summing at the confluence points. When using this method detention basins may be defined at confluence points in order to determine the effect of storage on the computations.

All of the computations for peak flows, hydrographs, and routing are done within *WMS*.

13.2 Rational Method Equation

The rational method is used around the world for peak flow estimation of small rural drainage basins and is the most widely used method for urban drainage design. The rational method equation is given in Equation 13.1.

$$Q = kCiA \quad \text{13.1}$$

where

- Q - peak flow (cfs or m^3/s).
- k - conversion factor equal to 1.008 (SI) or .00278 (metric).
- C - dimensionless runoff coefficient.
- i - rainfall intensity (in/hr, mm/hr).
- A - catchment area (acres, ha).

The rational method is generally considered to be an approximate model for computing the flood peak resulting from a given rainfall, with the runoff coefficient accounting for all differences between the rainfall intensity and the flood peak. Such differences result from infiltration, temporary storage, and other losses. A table of C values recommended by the American Society of Civil Engineers and Water Pollution Control Federation is shown below.

Area Description	**Runoff Coefficient C**
Business	
Downtown	0.70-0.95
Neighborhood	0.50-0.70
Residential	
Single-Family	0.30-0.50
Multiunits, detached	0.40-0.60
Multiunits, attached	0.60-0.75
Residential (suburban)	0.25-0.40
Apartment	0.50-0.70
Industrial	
Light	0.50-0.80
Heavy	0.60-0.90
Parks, cemeteries	0.10-0.25
Playgrounds	0.20-0.35
Railroad yard	0.20-0.35
Unimproved	0.10-0.30
Character of surface	**Runoff Coefficient C**
Pavement	
Asphaltic and concrete	0.70-0.95
Brick	0.70-0.85
Roofs	0.75-0.95
Lawns, sandy soil	
Flat, 2 percent	0.05-0.10
Average, 2-7 percent	0.10-0.15
Steep, 7 percent	0.15-0.20
Lawns, heavy soil	
Flat, 2 percent	0.13-0.17
Average, 2-7 percent	0.18-0.22
Steep, 7 percent	0.25-0.35

Equation 13.1 was developed from a simplified analysis of runoff using isochrones, or lines of equal travel time with areas in acres between them as illustrated in Figure 13-1. The method assumes no temporary storage in the basin and so the ratio between the peak runoff and the rainfall intensity is then the same as the ratio of the volumes of runoff and rainfall. If a constant rainfall intensity (in/hr) begins at time t=0 and has a duration of the time of concentration (t_c) for the basin, the hydrograph will reach an instantaneous peak at Ci. The t_c of the basin can be thought of as the time after rainfall excess begins to when all portions of the watershed are contributing to the peak flow at the outlet. If the duration is longer than t_c, the hydrograph will remain constant after reaching a value of Ci for a time period equal to the difference of the rainfall duration and t_c. In either case the time of rise and time of recession are equal to t_c.

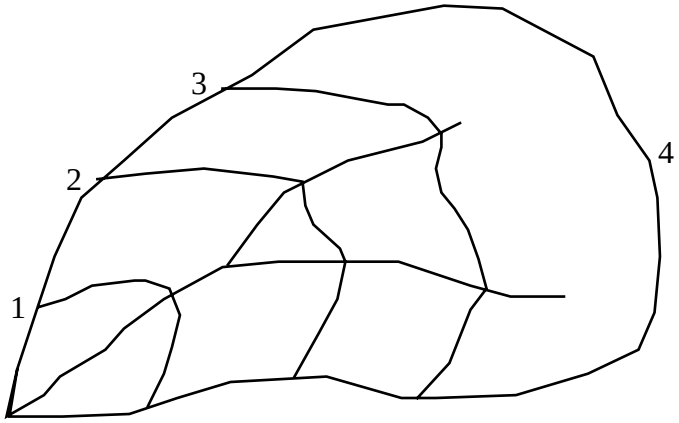


Figure 13-1 Basin Model Used to Develop Rational Method Formula.

With such a derivation, the effects of rainfall and basin size are accounted for explicitly, while most other physical characteristics of the watershed are accounted for indirectly by the time of concentration and runoff coefficient. This simple equation illustrates the critical nature of t_c. For durations less than t_c, the entire area is not contributing. For durations larger than t_c there is no increase in contributing area, and therefore no increase in peak flow.

13.2.1 Important Limitations

Due to assumptions regarding homogeneity of rainfall and equilibrium conditions at the time of peak flow, the rational method should not be used on areas larger than about 1 mi^2 without subdividing the overall watershed into subbasins including the effect of routing through any drainage channels.

As described in the introduction, *WMS* includes two different methods for determining runoff from larger watersheds, subdivided into smaller sub-

basins, including the ability to account for routing and lag through drainage channels and detention basins.

13.3 Computing Peak Flows with the Rational Method Equation

Like the other models supported by *WMS*, the Rational Method can be defined for a watershed/catchment developed from feature objects, DEMs, or TINs, or built using the tools provided in the hydrologic modeling module under the *Tree* menu. Once the topologic tree has been constructed the *Rational Method* (Figure 13-2) dialog can be accessed using the *Run Simulation* command from the *Rational* menu.

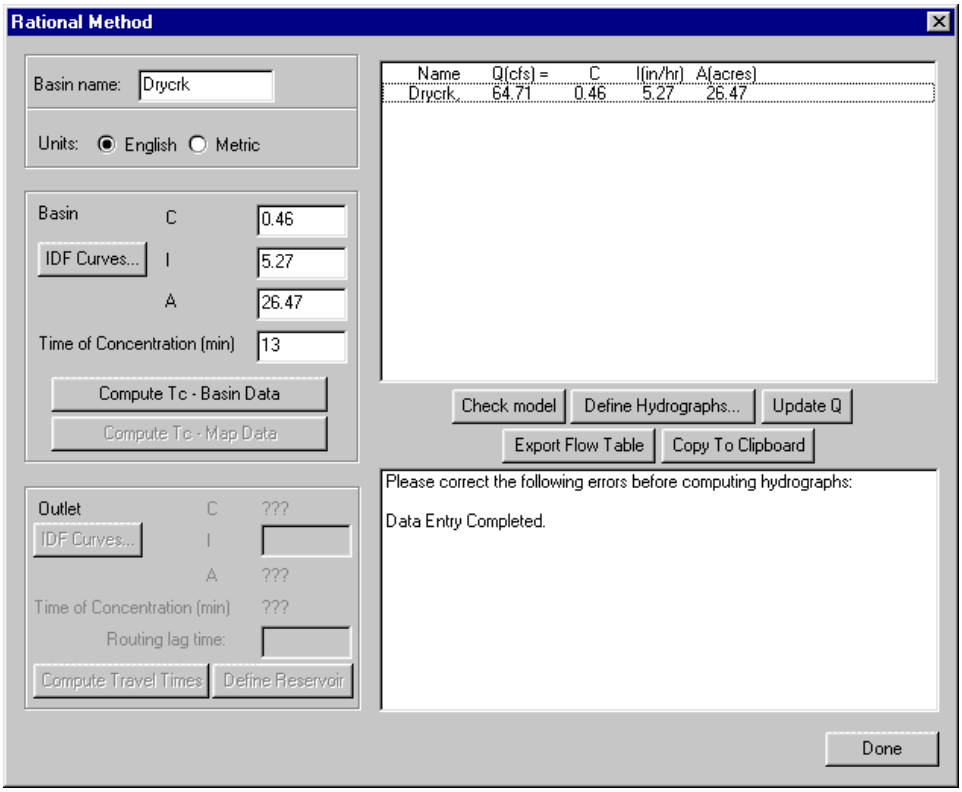


Figure 13-2 The Rational Method Dialog.

The *Rational Method* dialog allows you to enter all of the necessary values for computing a peak flow for a selected catchment area, or confluence point. The values for C, i, and A represent the values of the currently selected basin or outlet.

13.3.1 Editing Basin Data

If multiple basins are selected while the dialog is opened, the edit fields are grayed out (not dimmed) and follow the rules of multi-selection. For example

if you wish to set the value of i for all basins then you can select all basins, open the *Rational Method* dialog, click once on the edit field for i and specify the value. C could be set in a similar fashion, while A cannot be changed when multiple basins are selected.

All values used for computing peak flow with the rational method are stored with the basin attributes so that each time you select a basin the edit fields are filled with the values of that basin. This also makes it possible to use the areas computed from a feature object, DEM, or TIN using the *Compute Basin Data* command in the TIN, or DEM *Drainage* menu, or the *Update Basin Data* command in the *Feature Objects* menu.

Peak flow (Q) for each basin should update automatically, however the Update Q button may be used at any time to signify that you are finished entering the value for a given parameter (alternatively you can tab to the next field, or click in another edit field).

Peak flow values for all selected basins will be computed and displayed in the main text window of the Rational Method dialog.

The time of concentration, which is used to determine an appropriate rainfall intensity (i), can be determined in one of four ways:

1. The time of concentration can be computed "outside" of *WMS* and entered into the appropriate edit field.

2. Second the *Compute Tc – Basin Data* button can be chosen and one of the time of concentration (or lag time) equations specified (this option is only available when you have computed basin data from either a TIN or a DEM). The available equations in *WMS* are described in section 7.7.11.

3. A series of time computation arcs may be used to define overland, sheet, and channel flow within a basin and then travel times for each arc are summed to compute the total travel time or time of concentration for the basin. The *Compute Tc – Map Data* button accesses the dialog that allows you to combine arcs within the currently selected basin to compute a time of concentration. A description of the time computation coverage can be found in section 3.2.6 and it's use is described in more detail in section 15.3.

4. Finally the kinematic wave equation can be used from within the IDF curves dialog. See section 13.3.3 below for more information on this equation and how it is used in *WMS*.

13.3.2 Editing Outlet Data

Some of the information for outlet points is automatically determined from the contributing area upstream, while other parameters are entered separately. The

area is determined by summing the area of all upstream sub-basins. The runoff coefficient is computed from the upstream basins using the area-weighted equation shown below.

$$C_{OUTLET} = \frac{\sum A_i C_{BASINi}}{A} \quad \text{..... 13.2}$$

where:

C_{OUTLET} - The runoff coefficient for the outlet.

C_{BASINi} - The runoff coefficient for the i^{th} upstream basin.

Ai = The area of the i^{th} upstream basin.

A = The total upstream area at the outlet/confluence.

The time of concentration is determined by computing the longest combination of upstream time of concentration and channel travel time to the given outlet point. For example if a given outlet point had two contributing sub-basins the time of concentration for the outlet would be the longest time of concentration of the two upstream sub-basins. If there were other outlets upstream of the given outlet then the travel time would be added, and again the longest time or combination of times would be used as the time of concentration for the outlet.

The rainfall intensity value should be supplied separately for the outlet in the same way it is for a sub-basin. However, it is a function of the time of concentration and can be determined from an IDF curve relationship.

There are two ways routing of a hydrograph can be accounted for using the *WMS* implementation of the rational method. The first is to simply apply a time of travel between outlets. When hydrographs are computed (only with the summing method) at downstream outlets they are lagged by the travel time and added with other contributing basins. In addition simple level-pool reservoir routing may be performed on an outlet hydrograph before it is routed downstream (see section 15.6 for a complete description of the detention basin options available in *WMS* and included as an option in the rational method computations). Again, both of these options are available only when choosing the summing method of hydrograph generation rather than the traditional method where a peak flow and resulting hydrograph are determined from the time of concentration (and therefore rainfall intensity) at the outlet point.

13.3.3 Determining Rainfall Intensities

Precipitation intensity-duration-frequency (IDF) information is necessary for the specific locality in which the Rational Method will be used. In general this is done using either HYDRO-35 or NOAA Atlas 2 data. Using the *IDF*

Curves… button (for either basins or outlets), *WMS* can be used to develop curves from either of these two data sources, statistically derived data, or you can directly enter an i value if you typically compute it in another way or already know the design value you want to use.

The *IDF Computation* dialog (shown in Figure 13-3) can be used to create a series of T-year IDF curves from HYDRO-35, NOAA Atlas 2, or user defined data.

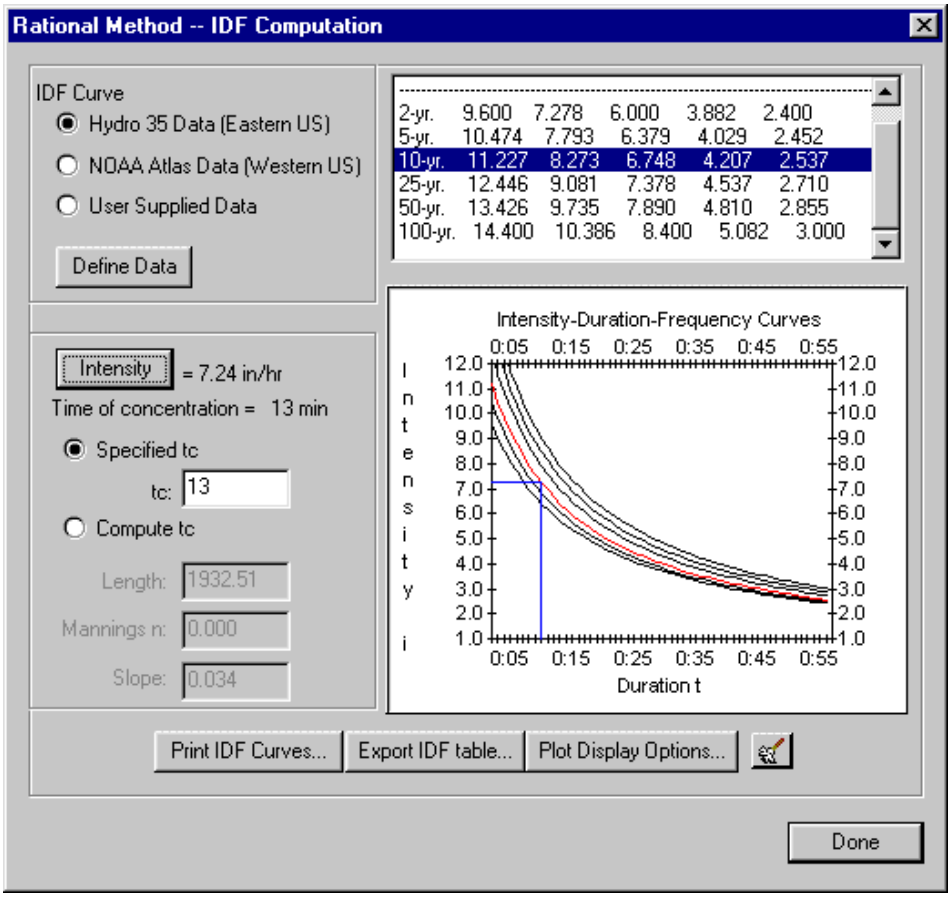


Figure 13-3 IDF Computation Dialog.

The type of data that will be used to create the IDF curves is specified with the radio group options in the upper left portion of the dialog. The *Define Data* button can then be used to bring a dialog which allows you to enter the following values, depending on the data type specified:

- **HYDRO-35** - The 2-yr 5, 15, and 60 minute rainfall values, and the 100-yr 5, 15, and 60 minute rainfall values. The HYDRO-35 maps for determining the six required rainfall values have been included at the end of this chapter.

- **NOAA Atlas 2** - The 2-yr 6 and 24 hour rainfall values, the 100-yr 6 and 24 hr rainfall values and the mean basin elevation.

- **User Defined** - The recurrence interval, and the 5, 10, 15, 30 and 60 minute rainfall values. With this option only a single curve for the recurrence interval will be generated.

Once the specified data has been entered the corresponding IDF-curves will be generated and plotted in the graphics window of the *IDF Computation* dialog. You can then specify the recurrence interval you want to use for analysis by selecting it in the text list window in the upper right portion of the dialog. After selecting the recurrence interval, the appropriate curve in the plot window will be displayed in red. Finally, an intensity value, i, is determined by specifying a time of concentration as outlined below and then clicking on the *Intensity* button. When clicking on the *Done* button for the *IDF Computation* dialog, the computed intensity value will automatically be updated in the edit field for i of the *Rational Method* dialog.

Computing the Time of Concentration

A time of concentration value needs to be entered in order to determine the intensity value to be used in the Rational Method equation. This value can either be entered manually or computed from the overland flow length, Manning's n, and slope. One equation used to compute the time of concentration from basin geometric parameters is the kinematic wave equation:

$$t_c = \frac{KL^{0.6}n^{0.6}}{i^{0.4}S^{0.3}} \quad \text{..... 13.3}$$

where:

t_c = the time of concentration.

L = overland flow length.

n = Manning's roughness coefficient.

i = rainfall intensity.

S = average slope of the overland area.

K = .93

This method has been adopted by the FHWA for general use, but other equations can be used as defined earlier in this chapter.

13.3.4 Computing Hydrographs with the Rational Method

The Rational Method equation is designed to compute peak flows. However a hydrograph, based on the peak flow and basin (or outlet) time of concentration, can be computed using one of five different unit hydrographs. Furthermore, there are two different methods (traditional and route by summing) hydrographs can be computed at confluence points.

The *Rational Method Hydrographs* dialog (Figure 13-4) is used to specify the dimensionless unit hydrograph method, and the way hydrographs at outlets are computed.

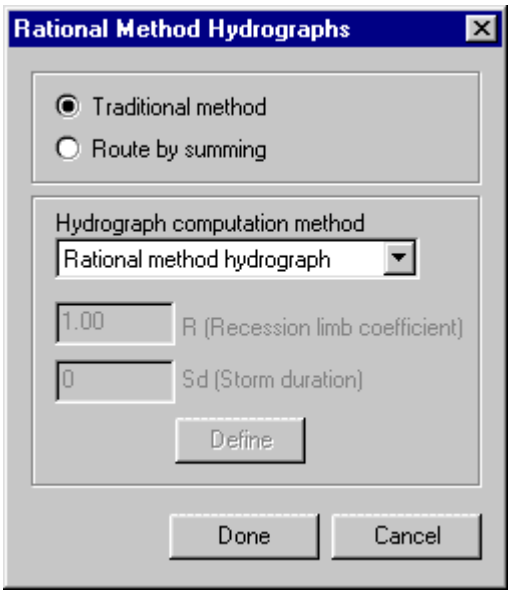


Figure 13-4 Rational Method Hydrographs Dialog.

Traditional vs. Route by Summing Methods of Outlet Hydrograph Computation

As stated before, the main difference between the two methods is that the traditional method computes a peak flow at an outlet by determining composite C, t_c, and areas from upstream basins and tributaries. Using the composite tc a rainfall intensity from an IDF curve is chosen to compute peak flow and then one of the hydrograph methods is used to develop a runoff hydrograph. The route by summing method on the other hands computes peak flows and hydrographs for basins in the traditional manner, but hydrographs at outlet points are determined by combining or summing the hydrographs from upstream basins and tributaries. The lag time determines the timing offset for hydrographs that "arrive" at an outlet through a tributary channel. With this method you also have the option of defining a detention basin, through which the outlet hydrograph may be passed. The detention basin is defined and calculated using the same techniques as outlined in the hydrologic calculators (see section 15.6).

The differences can probably best be understood with the following example. Figure 13-5 shows a diagram of the basin containing two "upstream" basins that combine at a junction and are routed through a lower basin. The areas, runoff coefficients, times of concentration and lag times for each basin and for the outlets (using the traditional method) are as shown.

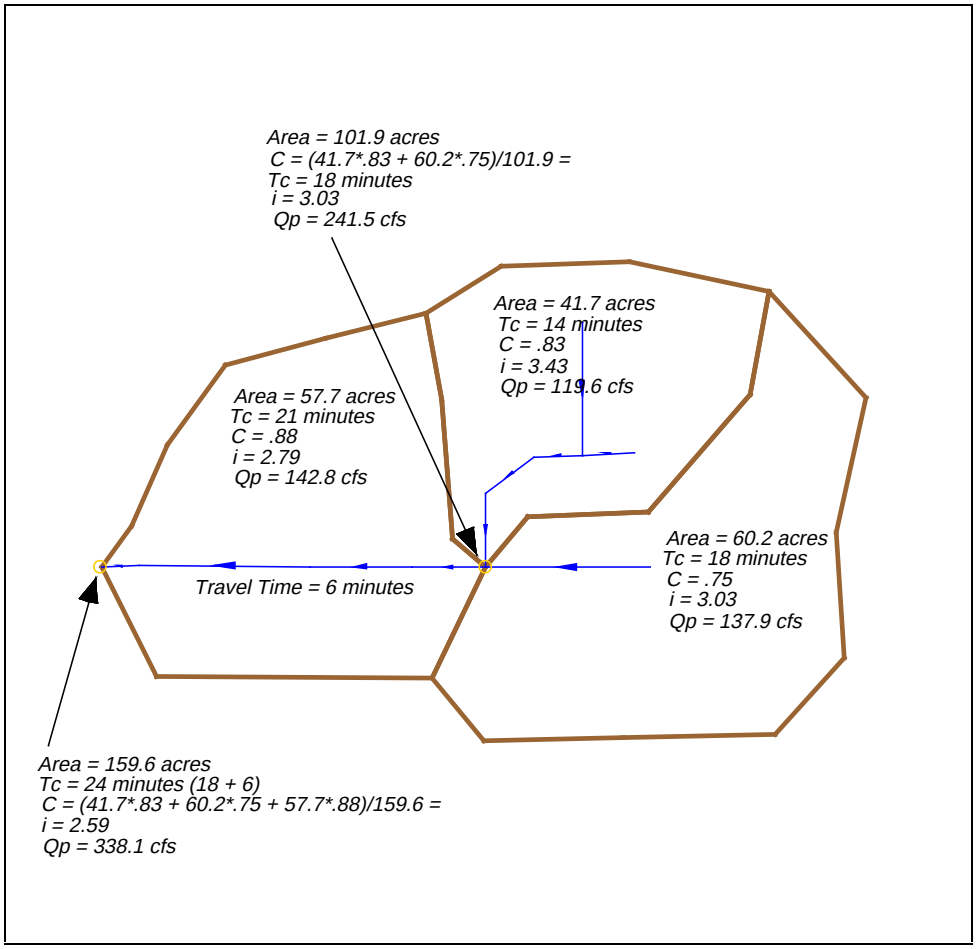

Area = 101.9 acres
C = (41.7*.83 + 60.2*.75)/101.9 =
Tc = 18 minutes
i = 3.03
Qp = 241.5 cfs
Area = 41.7 acres
Tc = 14 minutes
C = .83
i = 3.43
Qp = 119.6 cfs
Area = 57.7 acres
Tc = 21 minutes
C = .88
i = 2.79
Qp = 142.8 cfs
Area = 60.2 acres
Tc = 18 minutes
C = .75
i = 3.03
Qp = 137.9 cfs
Travel Time = 6 minutes
Area = 159.6 acres
Tc = 24 minutes (18 + 6)
C = (41.7*.83 + 60.2*.75 + 57.7*.88)/159.6 =
i = 2.59
Qp = 338.1 cfs

Figure 13-5 Sample Rational Method Model.

Rainfall intensities are determined using the times of concentration and an IDF curve. As would be expected the higher the time of concentration, the smaller the intensity.

Peak values for the three sub-basins are identical for each of the two methods and are shown in Figure 13-5. However, for the outlet points the peak flows (and therefore hydrographs) are different.

In the case of the traditional method the time of concentration for the "upstream" outlet (the location where the two upper basins join) is determined to be 18 minutes (the largest tc of the two basins). From this t_c a rainfall intensity of 3.03 is determined from the IDF curve. The contributing area is

101.9 acres (the sum of the two upstream basins), which when using equation 13.1 results in a peak flow of 241.7 cfs.

For the route by summing method the two upstream hydrographs are summed and the resulting peak flow value is determined. In our example the standard rational method (triangular shaped hydrograph) option was chosen. Since the times of concentration are different (and therefore the times to peak for the hydrographs) the resulting peak flow is not the sum of the two basin peak flows, but as can be seen in Figure 13-6, is 225.

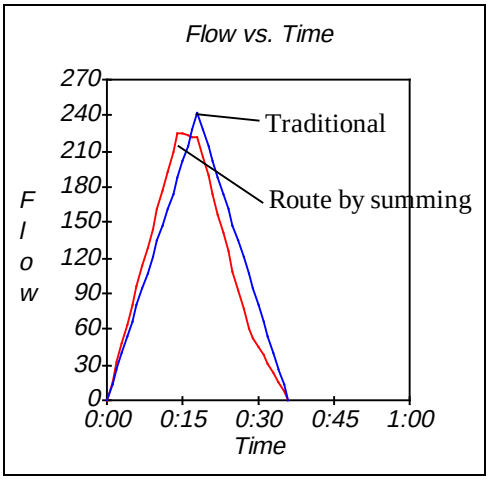


Figure 13-6 Rational Method Hydrograph Comparison Upper Outlet.

For the lower outlet point a similar comparison can be made. The only difference is that the time of concentration is the longest time between the time of concentration of the lower basin (21 minutes), and the longest time of concentration of the two upstream basins plus the travel time through the tributary (18 + 6 = 24 minutes). In this case it is 24 minutes. Using the traditional method the rainfall intensity for a time of concentration equal to 24 minutes is 2.59, the total summed area is 159.6 acres and the resulting peak flow 338.1 cfs. The peak flow for the route by summing methods is 366, and the comparison of hydrographs using the standard triangular rational method dimensionless unit hydrograph is shown in Figure 13-7.

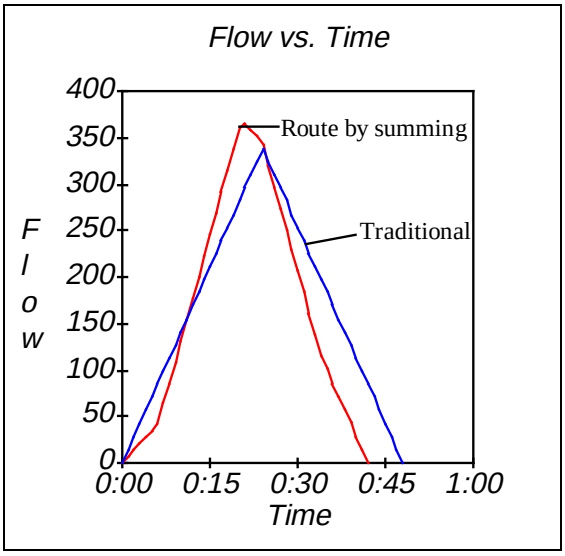


Figure 13-7 Rational Method Hydrograph Comparison Lower Outlet

One final difference is that with the route by summing method the addition of reservoirs (detention basins) in the calculations is possible. However, with either method a resulting hydrograph could be used in the design of a detention basin as a separate operation (see section 15.6).

Rational and Modified Rational Unit Hydrograph

The Rational Unit Hydrograph has a time to peak equal to the time of concentration. Both the rising and receding limbs of the hydrograph have a duration equal to the time of concentration, and therefore the shape of the hydrograph is an isosceles triangle with a time base of $2*t_c$ as shown in Figure 13-8. This method is chosen by specifying the *Modified Hydrograph Method* using 1.0 for the recession limb coefficient and t_c for the storm duration.

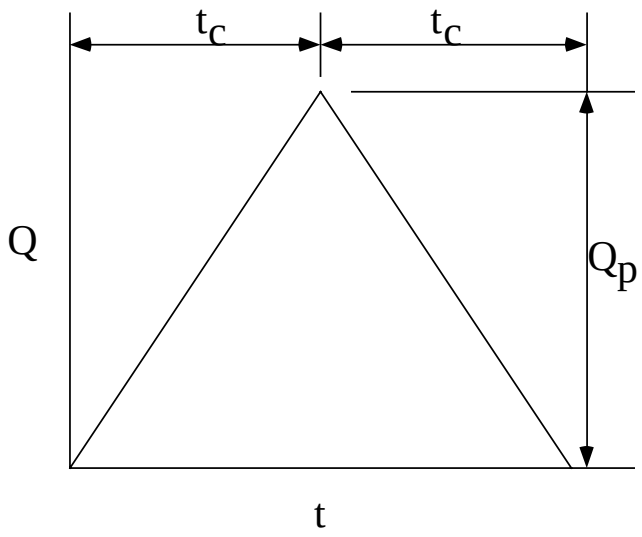


Figure 13-8 Rational Unit Hydrograph

The Modified Rational Hydrograph also assumes that the time to peak is equal to the t_c, but allows for the duration of the storm to be longer than t_c, resulting in a trapezoidal shaped hydrograph as shown in Figure 13-9. A coefficient to modify the slope of the receding limb may also be applied with this method.

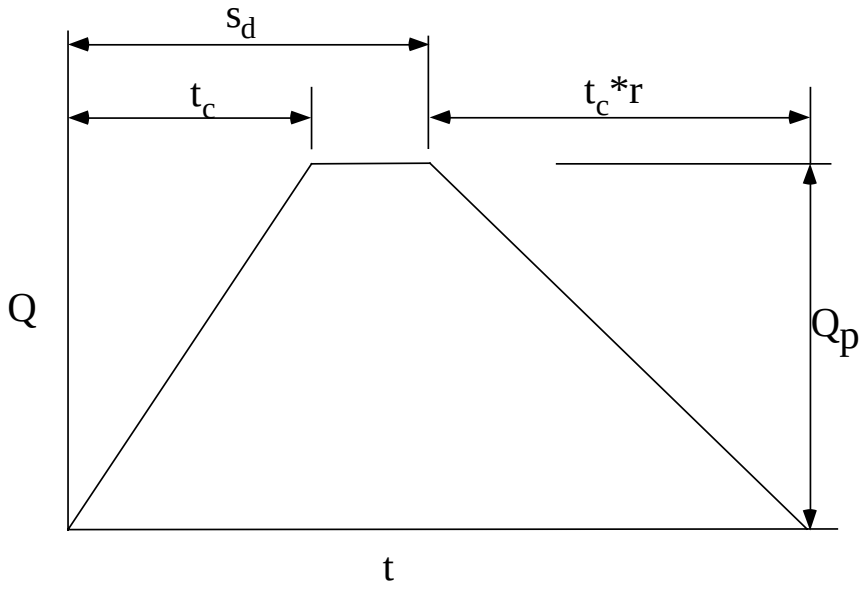


Figure 13-9 Modified Rational Hydrograph

where:

t_c = time of concentration.
Q = Flow at time t, in cfs.
Q_p = Peak flow.

r = falling limb coefficient

s_d = storm duration

Universal Rational Hydrograph

The Universal Rational Hydrograph uses a set of coefficients and the peak discharge to compute the hydrograph ordinates at different times. The coefficients are shown in the following table and the resulting hydrograph is displayed in Figure 13-10.

Dimensionless Time and Hydrograph Ordinates

t/t_c	Q/Q_p
0	0.00
1	0.21
2	0.30
3	1.00
4	0.54
5	0.39
6	0.25
7	0.18
8	0.15
9	0.14
10	0.13
11	0.00

Notice that the peak occurs at $3t_c$, and the time base is equal to $11t_c$.

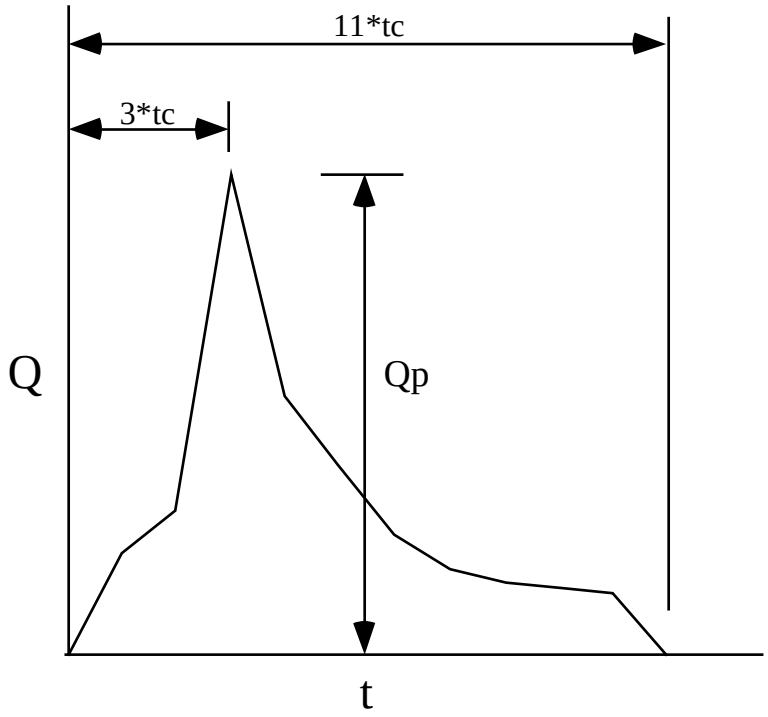


Figure 13-10 Universal Rational Hydrograph

where:

t_c = time of concentration.

Q = Flow at time t, in cfs.

Q_p = Peak flow.

Dekalb Rational Hydrograph

The Dekalb Rational Hydrograph was developed by Dekalb County Georgia, and like the Universal Rational Hydrograph, ordinates are computed by scaling the peak discharge by an appropriate value. The time to peak occurs at $5t_c$, while the time base is $10t_c$. All coefficients occur at increments of t_c, and are different depending on whether t_c is less than 20 minutes or not. The following table lists the coefficients and a typical hydrograph is shown in Figure 13-11.

Dimensionless Time and Hydrograph Ordinates

t/t_c	Q/Q_p for tc < 20 min	Q/Q_p for tc ≥ 20 min
0	0.00	0.00
1	0.16	0.04
2	0.19	0.08
3	0.27	0.16
4	0.34	0.32
5	1.00	1.00
6	0.45	0.30
7	0.27	0.11
8	0.19	0.05

9	0.12	0.03
10	0.00	0.00

where:

t_c = time of concentration.

Q = Flow at time t, in cfs.

Q_p = Peak flow.

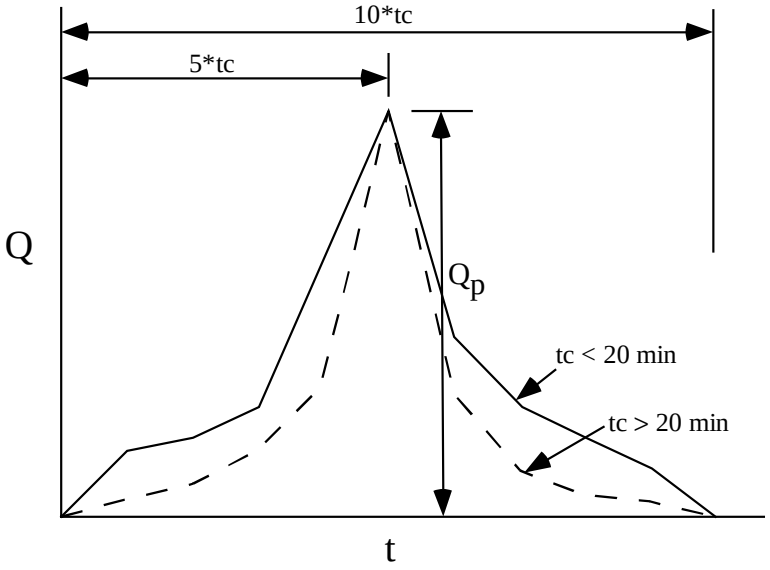


Figure 13-11 Dekalb Rational Hydrograph.

User-Defined Hydrograph

With the *User Defined* method of hydrograph generation, you specify the number of ordinates for a dimensionless unit hydrograph and then define the t/t_c and Q/Q_p values. NOTE: All t/t_c values must be even integers.

User defined files may be exported or imported so that they do not need to be re-entered from one run of *WMS* to the next.

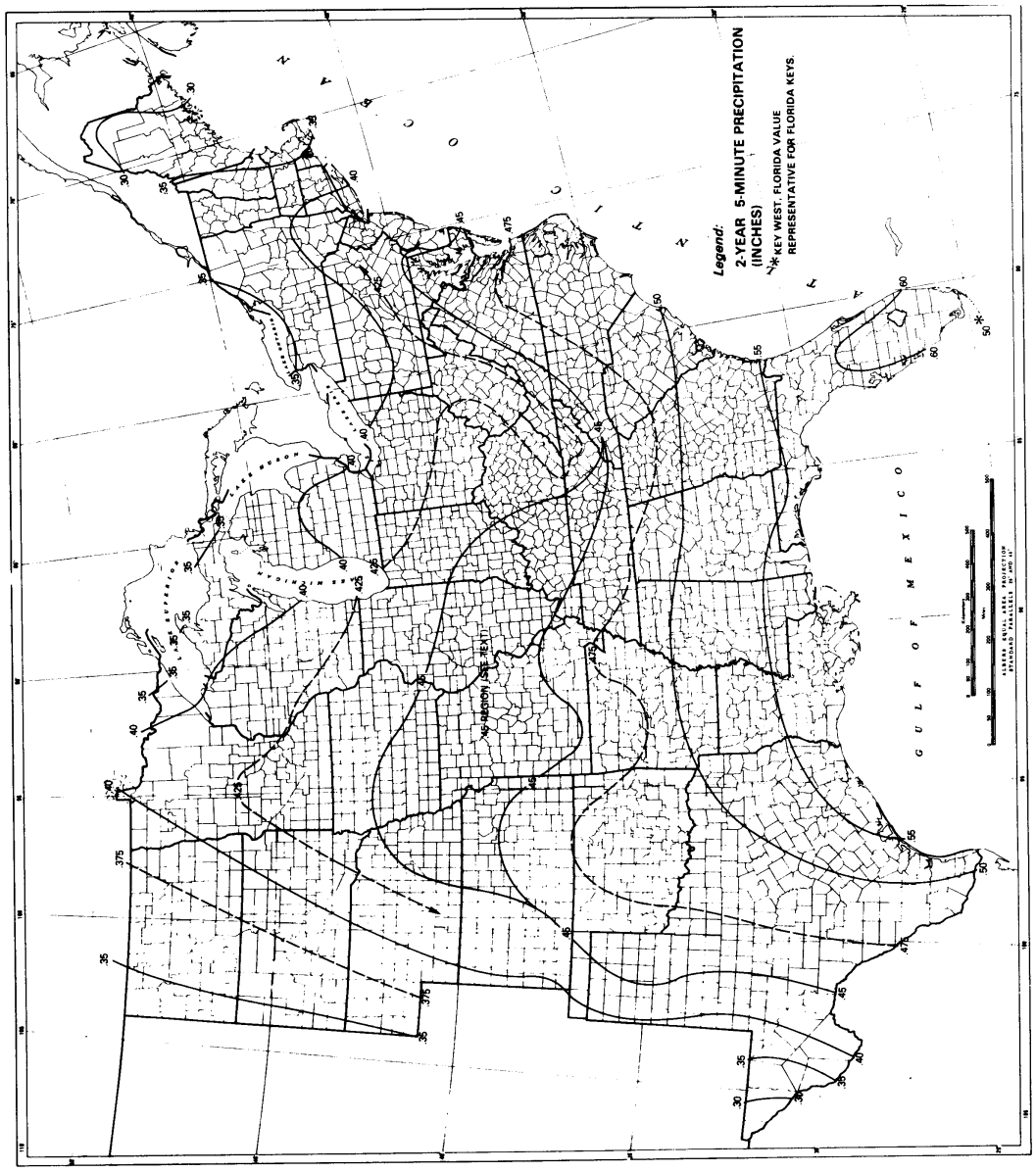
13.4 Saving and Restoring a Simulation

The *Save Simulation* command in the *Rational* menu can be used to save the topologic tree structure with any rational method parameters which have been defined. The *Read Simulation* command will restore the tree and parameters so that you may continue with a particular model.

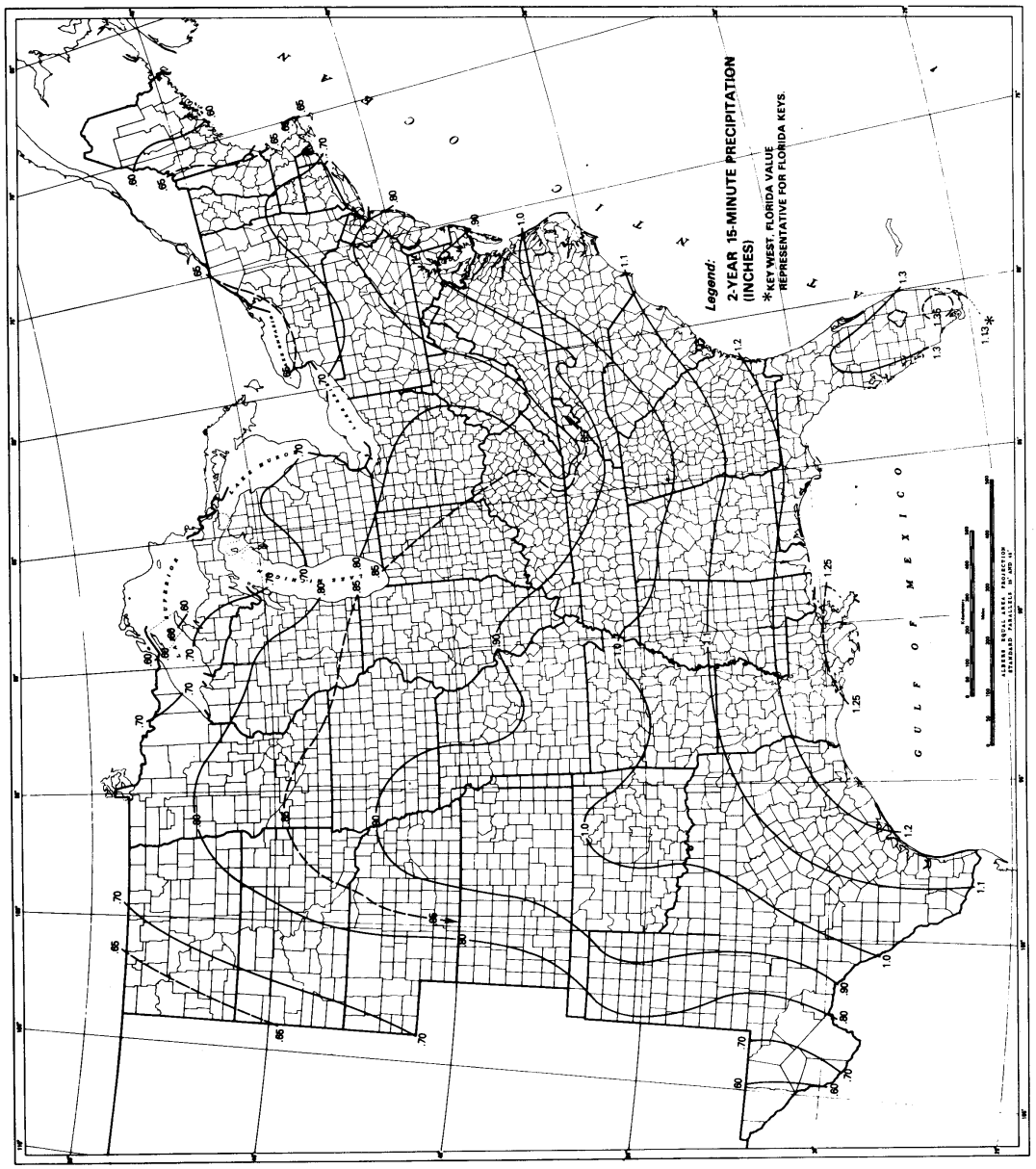
13.5 MODRAT

The *Rational* menu also contains several options to create and run a MODRAT simulation. MODRAT is a modified rational method developed and used by the Los Angeles Department of Public Works. Given the limited scope of applicability of MODRAT, this reference manual does not cover the use of the MODRAT interface. Information on the use of the MODRAT interface as well as the MODRAT model may be obtained by contacting your distributor of WMS or the Los Angeles County Department of Public Works.

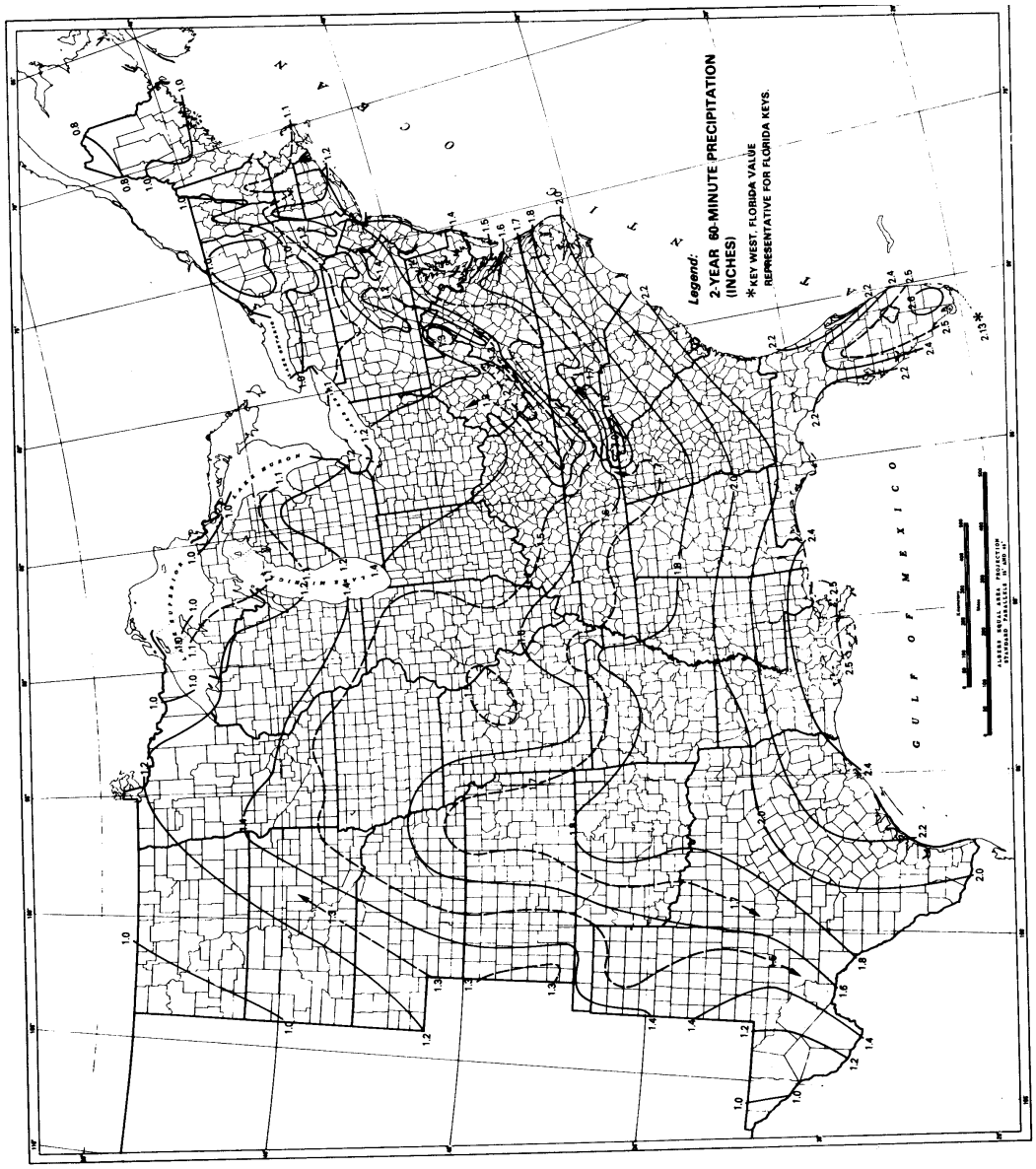
13.6 2-year 5-minute Precipitation (HYDRO-35)


Legend:
2-YEAR 5-MINUTE PRECIPITATION (INCHES)
KEY WEST, FLORIDA VALUE REPRESENTATIVE FOR FLORIDA KEYS.
GULF OF MEXICO
NO RADIOW (SEE TEXT)

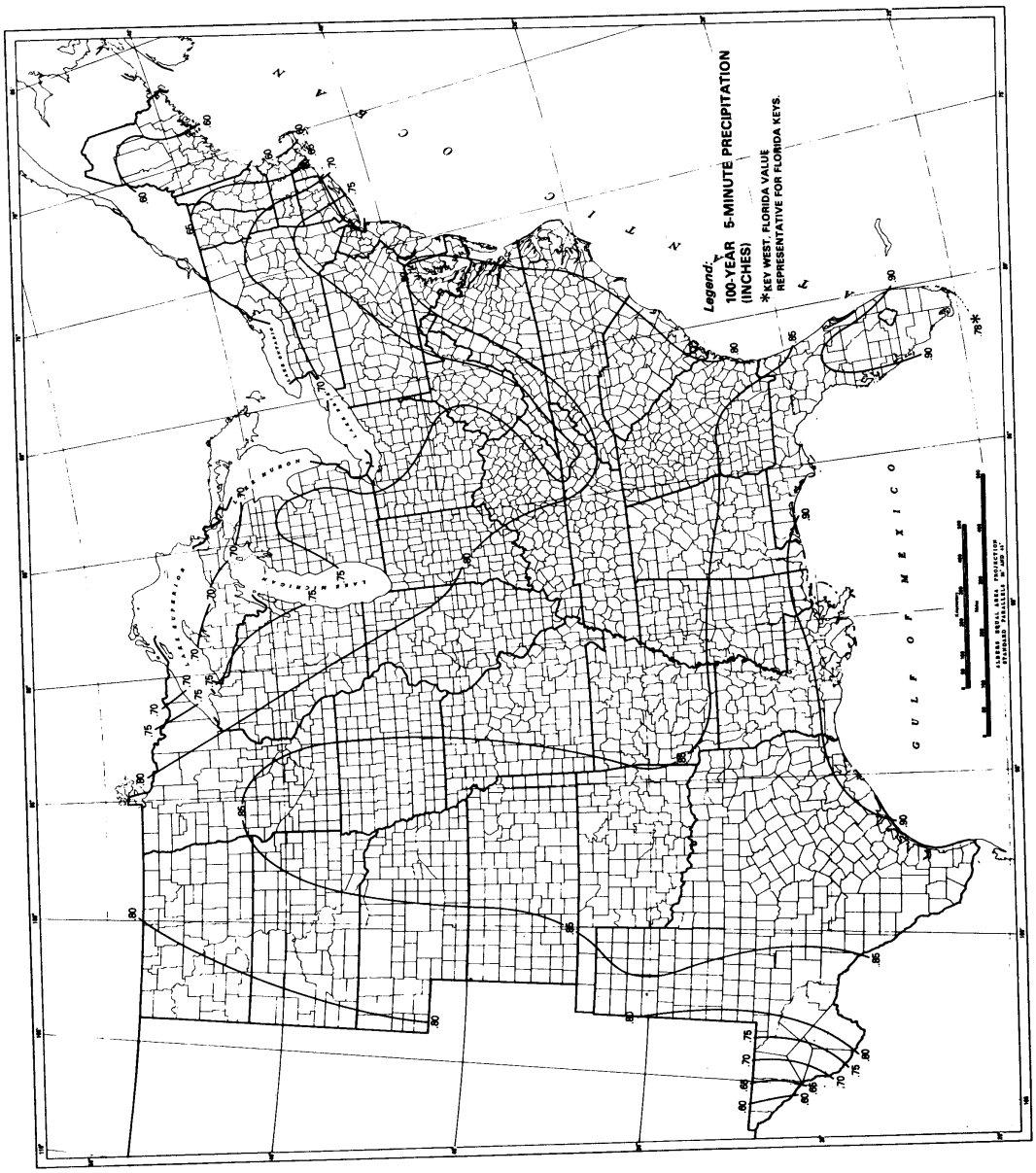
13.7 2-year, 15-minute Precipitation (HYDRO-35)



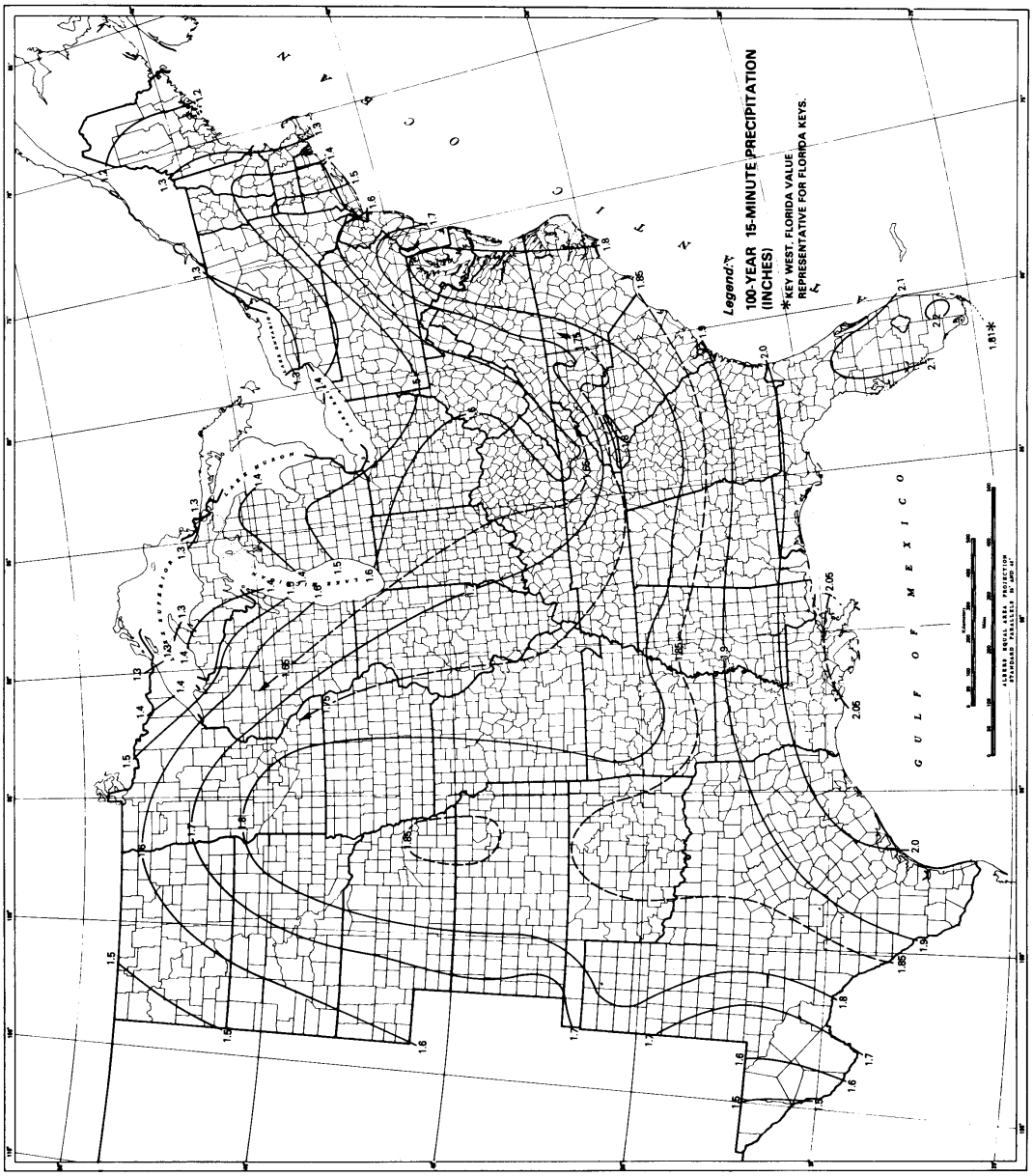
13.8 2-year, 60-minute Precipitation (HYDRO-35)



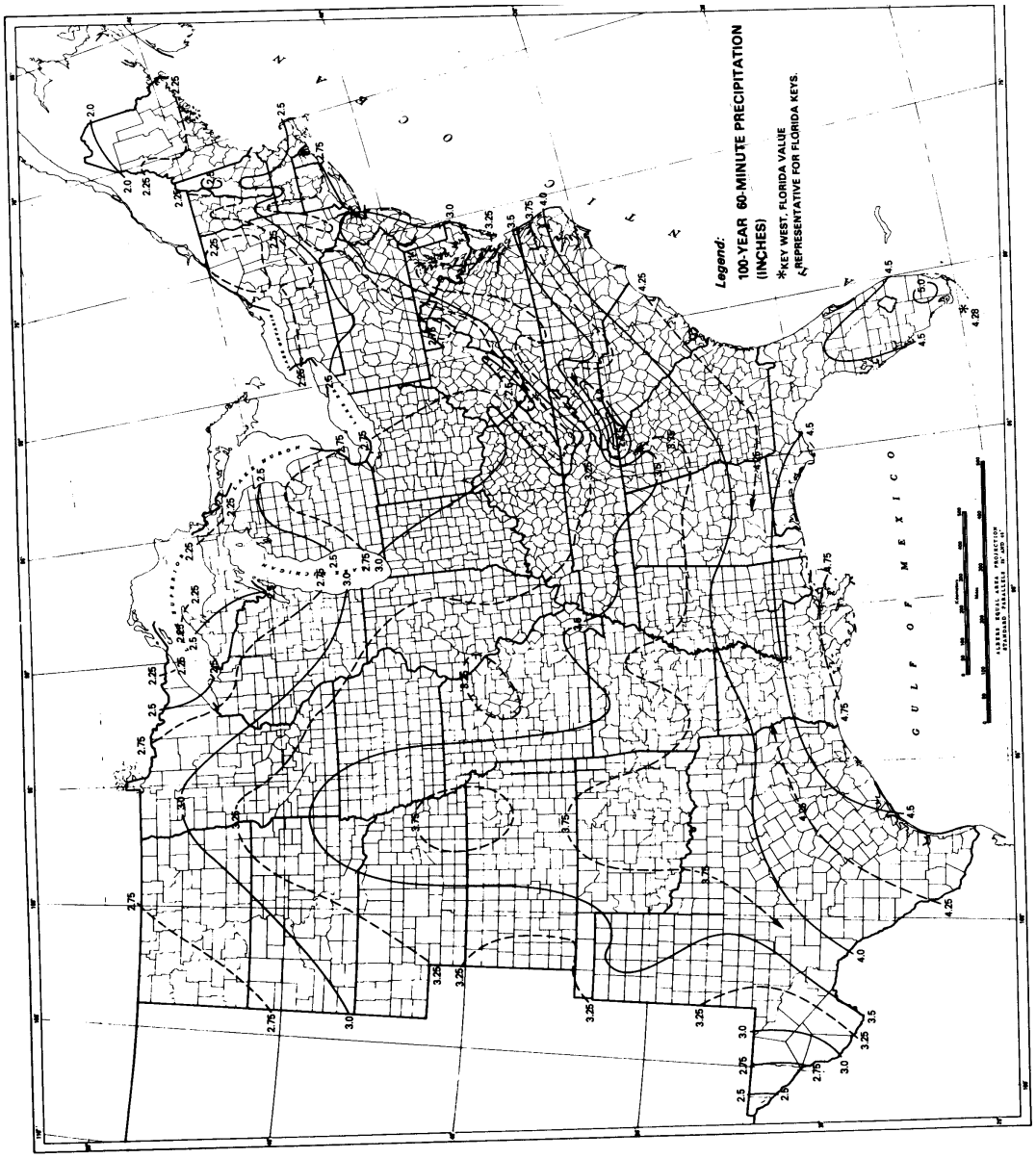
13.9 100-year, 5-minute Precipitation (HYDRO-35)



13.10 100-year, 15-minute Precipitation (HYDRO-35)



13.11 100-year, 60-minute Precipitation (HYDRO-35)



CHAPTER 14

National Flood Frequency Program

14.1 Introduction

WMS includes an interface to the National Flood Frequency Program (NFF). The NFF program is a compilation of all the current (as of September 1993) statewide and metropolitan area regression equations. The regression equations are a result of years of effort by the United States Geological Survey (USGS) to develop regional regression equations for estimating flood magnitude and frequency of ungaged watersheds. The USGS, in cooperation with the Federal Highway Administration and the Federal Emergency Management Agency compiled all the regression equations into a single database file. This database file is the basis of the NFF program, which can be used to guide the user through the input required to compute peak flows for different frequencies using the database of state by state regression equations.

The NFF interface in *WMS* provides a windows based, graphical user interface to the same database of regression equations. The entire program is run from a single dialog. Further, if a digital terrain model is available for the study area, all of the geometric parameters required for the regression equations are automatically supplied as the individual equations are specified. These parameters include area, slope, elevation, distances, and others.

The NFF equations are useful for estimating a peak flood discharge and typical flood hydrograph for a given recurrence interval of an unregulated rural or urban watershed. These techniques should be useful to engineers and hydrologists for planning and design purposes. A statewide summary, along with other technical information can be found in the USGS Water-Resources Investigations Report 94-4002. The rest of this chapter describes the interface

to the NFF program. You should refer to the USGS report for technical descriptions on the methods used by NFF for computing peak flows and hydrographs, and for state by state information.

14.2 NFF Run Simulation Dialog

The NFF Run Simulation dialog controls all of the input required for peak flow estimations using the USGS state by state regression equations. The dialog is shown in Figure 14-1.

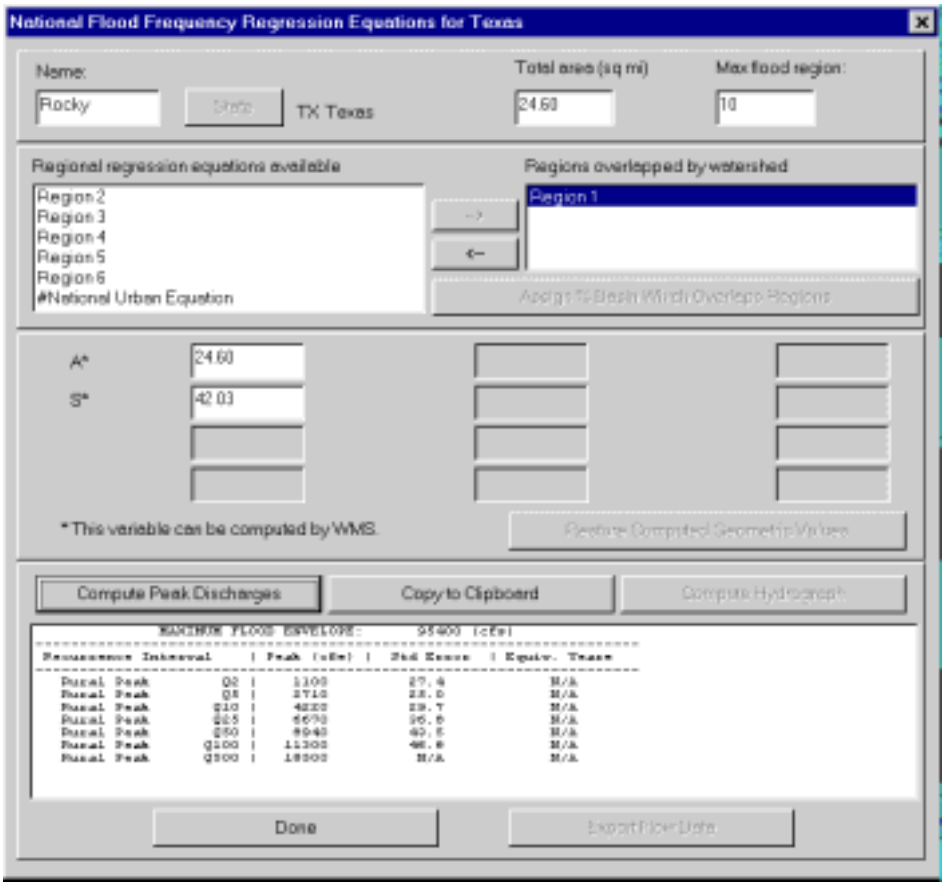


Figure 14-1 Run NFF Simulation Dialog.

The text window at the lower center of the dialog acts as both a model checker, leading you step by step in setting up a simulation, as well as the location where peak flow results are displayed. The following sections describe the steps necessary to set up a simulation.

14.2.1 Basin Name

The basin name is not required to run a simulation, and is primarily used for identification in the topologic tree window. If you are going to run either *HEC-1* or TR20 with the same model you should keep the name to six characters or less.

14.2.2 State

The NFF regression equations are separated by state. The *State* button is used to specify the state that your watershed is located in. Once the state is specified, the available regions will appear in the regional equations text window. Besides the 50 US states an equation for Puerto Rico and a custom defined equation can be selected from the available choices (see section 14.2.6).

14.2.3 Total Watershed Area

The total watershed area includes the area of all regions overlapped by the watershed. If a terrain model is used to compute areas, this value will be defaulted to the area computed for the selected basin.

14.2.4 Maximum Flood Region

For comparison and evaluation, the NFF program compares each extrapolated 500-year flood peak discharge with the maximum flood-envelope curves given by Crippen and Bue (1977) and Crippen (1982). The map at the end of this chapter shows the different flood regions, and should be used to determine the appropriate region for your watershed. A default value has been determined for each region, but this value may be overwritten. This figure is also included on page 16 of the USGS report 94-4002.

14.2.5 Regional Regression Equations

Each state is divided up into hydrologically similar regions with separate regression equations. Once the state is selected, the available regions appear in the text window below the name.

Selecting Overlapping Regions

You may select the region(s) the study watershed overlaps by clicking on the region name in the *Regional regression equations available* text window and then clicking on the arrow pointing to the right. The region named will be "moved" to the *Regions overlapped by watershed* text window, and any relevant parameters will be activated. If a mistake is made the regional equation can be "moved" back again using the button with an arrow pointing

to the left after selecting the region in the *Regions overlapped by watershed* window.

The USGS Report has a description and applicable maps for each state. These maps can be used to identify which regions are overlapped by your study watershed.

State and National Urban Equations

The regional equations for each state correspond to analysis for rural watersheds. However each state includes the national urban equation in its list of regions. The national urban equation (preceded by a # character) contains additional parameter values and must be used in conjunction with the appropriate regional rural equation. In addition, some states include regional urban equations (preceded by a $ character) which are used separately from the rural equations. The list of available regression equations changes according to which equations have already been selected. For example if a regional urban equation is selected all other equations disappear, since it must be used by itself. Further, if a regional rural equation is selected any regional urban equations are removed since they cannot be combined.

Assigning Percentage of Overlapped Area in Each Region

Most watersheds will likely fall within a single region. In such cases you will not need to worry about assigning percentages, since the area parameter for the selected region will receive 100% of the total area for the watershed. However, if a watershed does overlap two or more basins then the percentage of total area must be assigned to each.

When the first region is moved to the overlap window the area parameter is defaulted to the total area as assigned in the edit field at the top middle of the dialog. If this area is changed (to a smaller value), then when the second region is moved it's area value will be defaulted to the difference between the total area and the area assigned to the first region. These areas can be viewed/changed at any time by clicking on the *Assign % Watershed Which Overlaps Regions* button.

Assigning Regression Equation Parameters

As regions are selected in the *Regions overlapped by watershed* window the relevant parameters for the regional equations are activated in the edit fields. Any parameter with an * after the name is a parameter that is computed by *WMS* when drainage data for a basin is computed from a digital terrain model. These parameters are defaulted to the computed values at the time the regression equation is selected for use. You may over-ride defaulted values if you wish, and later you could return to the computed values by clicking on the *Restore Values from TIN* button.

The NFF program includes in its database the range of parameter values for the watersheds used in developing the regression equations. The applicable range is displayed in the help window whenever the cursor is in the parameter value edit field. Since the NFF Simulation dialog is large, you may need to move it so that it does not obscure the help window dialog.

14.2.6 Using a Custom Defined Equation

Because there is likely to be some lag between the time a new regression equation is made available and the time it is included in the NFF equations database compiled by the USGS, an option to import a custom defined equation in *WMS* is available. This makes it possible to use a regression equation for your state which has been changed/created since the time *WMS* last compiled the USGS database.

To select a custom equation you should select the *Custom* option from the *States* dialog. The custom equations are defined in an ASCII formatted file and by default *WMS* will automatically look for a file named *custnff.equ* in the same directory as the *WMS* executable file is located. If this file is not found then the file browser allows you to select a file from any location. The format, and instructions for setting this file up is documented in section 20.21.

14.2.7 Computing Peak Discharges

The text display window at the bottom of the dialog lists the steps required to compute peak discharges. These steps have been outlined in the previous sections. As each step is completed a "Done" is written in front of the step. Once all steps are completed the *Compute Peak Discharges* button can be used to estimate the peak flows for the 2, 5, 10, 25, 50, 100, and 500-year recurrence intervals (some equations do not include all recurrence intervals). The peak flow values are listed in the same text window where the steps for setting up models is given.

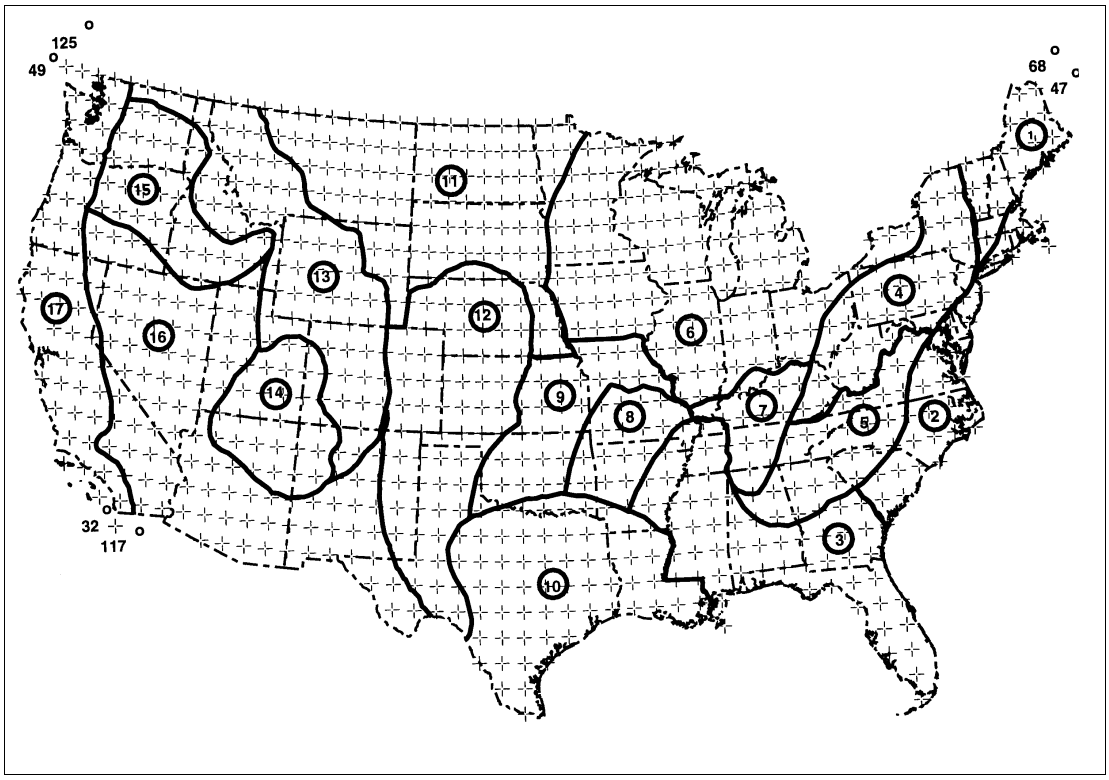
Creating Hydrographs

Once peak flows have been computed, a hydrograph can be generated by selecting the desired recurrence interval and clicking on the *Compute Hydrograph* button. The NFF program contains a procedure for computing a typical hydrograph that represents average runoff for a specified peak discharge. The method uses a dimensionless hydrograph method which needs as input the peak discharge, the basin lagtime, and the dimensionless hydrograph. Once the hydrograph has been computed it can be selected for display in the *Hydrograph* window or saved/listed using the *List* command in the *Hydrograph* menu. To get back to computed discharges, re-select the basin in the topological tree window and click on the *Compute Peak Discharges* button.

14.3 Saving and Restoring a Simulation

The *Save Simulation* command in the *NFF* menu can be used to save topologic tree structures with any state, regions and parameters which have been defined. The *Read Simulation* command will restore the tree and parameters so that you may continue with a particular model at a later time.

14.4 Maximum Flood Region Boundaries



CHAPTER 15

Hydrologic/Hydraulic Calculators

15.1 Introduction

Peak flows and/or hydrographs are often used to design important hydraulic structures such as channels, detention basins, culverts, etc. This chapter describes some of the design/analysis tools that can be used in combination with the runoff models supported by *WMS*. Some of these functions (curve number, storage capacity curves) are used to prepare input for the runoff models, while others use results of the runoff models to design hydraulic structures such as curb and gutter flow, weirs, detention basins, culverts, etc.

15.2 Computing SCS Curve Numbers, Runoff Coefficients, and Green & Ampt Infiltration Parameters

SCS curve numbers are typically determined by using a table relating land use to hydrologic soil type. The hydrologic soil type can be either A, B, C, or D, where the infiltration capacity decreases from A to D. The curve numbers for each soil group for a given land use are given in most books on hydrology. A composite curve number for a basin can be computed by taking an area-weighted average of the different curve numbers for the different regions (soil type and land use combinations) within a basin. The same thing can be done to compute a composite runoff coefficient, only in this case a table relating soil ID to runoff coefficient is used rather than a table for curve numbers.

Maricopa County, Arizona, and other regions often use the Green & Ampt infiltration options within *HEC-1*. Green & Ampt parameters may also be

mapped from soil type and land use coverages. Initial abstractions and percent impervious are determined from a land use coverage while a soil coverage is used to define hydraulic conductivity, soil moisture deficit, and wetting front suction. The parameters required to define these values must be entered for the appropriate coverages. See section 3.2.6 to learn more about defining parameters for soil and land use coverages.

WMS allows you to define a hydrologic soil coverage or grid and land use coverage or grid that define boundaries for the different soil types and land uses. Typically this information is imported from standard GIS formatted files (see section 2.8.6 for more information on importing these files). These data are then mapped to either drainage coverage polygons or TIN triangles (depending on how you have created your watershed and sub-basin boundaries) and used in the computation of a composite curve number. In summary the following data are used for computing composite CN's, runoff coefficients, or Green & Ampt parameters:

- Basin boundaries can be defined with either TINs or feature objects (remember that boundaries defined from a DEM are converted to feature objects).

- Land use IDs can be supplied from a land use coverage in the map module or as DEM (a gridded) attributes.

- Soil type IDs can be supplied from a soil type coverage in the map module or as DEM (a gridded) attributes.

- A user-defined table relating land use IDs to the parameters being mapped (for example SCS curve numbers, Green & Ampt parameters, etc.).

- A user-defined table relating soil IDs to the parameters being mapped (for example runoff coefficients, Green & Ampt parameters, etc.).

Any combination of the different data required for computation can be used (i.e. drainage coverage, land use grid, soil type coverage, etc.). If a land use or soil coverage is used then the parameters for each polygon ID can be defined using the *Attributes* command in the *Feature Objects* menu with the proper coverage being active (see page 3-14 for more information on assigning land use and other parameters to polygon IDs). However, if grid attributes are used for the soil or land use ID definitions then one way to define the parameters for each ID is by creating the mapping file with a text editor and then importing in the *Compute Composite Runoff Coefficients/CNs* dialog shown in Figure 15-1 below. The format for these mapping file types can be found in section 20.17.

Another way to define land use or soil type is to define a land use or soil type coverage. The *Attributes* command in the *Feature Objects* menu can then be used to assign id's to the individual polygons.

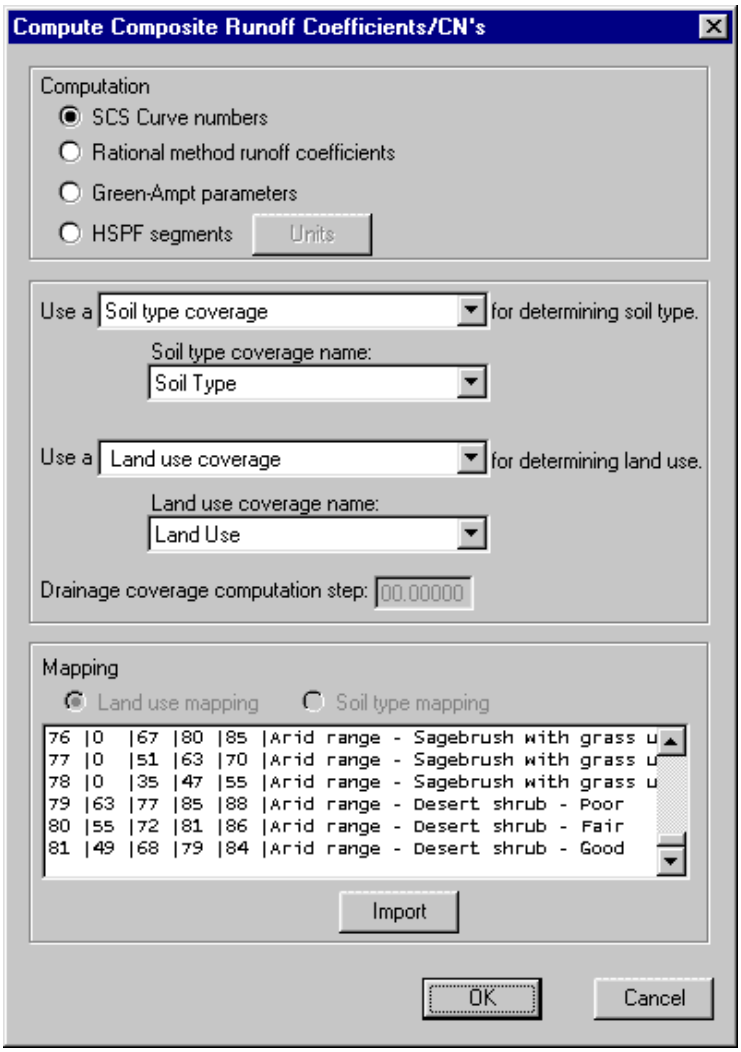

Compute Composite Runoff Coefficients/CN's
Computation
SCS Curve numbers
Rational method runoff coefficients
Green-Ampt parameters
HSPF segments
Units
Use a Soil type coverage for determining soil type.
Soil type coverage name:
Soil Type
Use a Land use coverage for determining land use.
Land use coverage name:
Land Use
Drainage coverage computation step: 00.00000
Mapping
Land use mapping
Soil type mapping
76 |0 |67 |80 |85 |Arid range - Sagebrush with grass u
77 |0 |51 |63 |70 |Arid range - Sagebrush with grass u
78 |0 |35 |47 |55 |Arid range - Sagebrush with grass u
79 |63 |77 |85 |88 |Arid range - Desert shrub - Poor
80 |55 |72 |81 |86 |Arid range - Desert shrub - Fair
81 |49 |68 |79 |84 |Arid range - Desert shrub - Good
Import
OK
Cancel

Figure 15-1 Compute Composite Runoff Coefficients/CN Dialog.

Once the polygon coverages and/or grid files for land use or soil types are defined, and the mapping tables set up, you are prepared to compute parameters for one of the available methods.

15.2.1 Computation Method

The computation method determines whether composite curve numbers, composite runoff coefficients, oGreen & Ampt infiltration parameters will be computed. This affects the type of mapping table that will be required and also where results are stored. When computing curve numbers the values are automatically stored with *HEC-1*, *TR-20*, *TR-55*, and any other model

requiring the use of a CN. If runoff coefficients are computed they are stored with the rational method data.

15.2.2 Soil Type

The soil type option determines whether a soil type coverage or a soil type grid will be used. The soil data has a slightly different meaning depending on whether CN or runoff coefficients will be computed. For CN the critical attribute is the hydrologic soil type (0-soil A, 1-soil B, 2-soil C, 3-soil D), whereas for runoff coefficients the critical attribute is a soil ID that can be related to a table of runoff coefficients. Additional instructions for defining soil type coverages are given on page 3-16.

15.2.3 Land Use

The land use option determines whether a land use coverage or a land use grid will be used. The critical attribute for land use is an ID that can be related to a table of parameters for curve numbers, Green & Ampt parameters, or HSPF segments. Additional instructions for defining land use coverages are given in section 3-14.

15.2.4 Computation Step

The computation step is only used when defining composite curve numbers or runoff coefficients for a drainage coverage. If a TIN is used then individual basins are composed of several triangles and each triangle can be assigned a land use and/or soil type. However for a drainage coverage each basin is typically represented by a single polygon. The computation step is used to divide each basin polygon into a number of square cells (the computation step being the length of a side) that are each assigned a land use and or soil type ID. The smaller the step length the more accurate the composite number will be, but the more time consuming the computation as well.

15.2.5 Mapping Table

The mapping table is different depending on whether composite curve numbers or runoff coefficients are being computed. A mapping table must be imported using the *Import* button prior to computing the composite curve numbers. The radio group above the mapping window specifies whether the land use or soil type mapping table is displayed. For a land use or soil type coverage the mapping table may be set up interactively by selecting each polygon and assigning the appropriate parameters. Such files can then be exported and later imported. For grid attributes you can define a land use or soil type table by defining a land use or soil type coverage and then using the Attributes command from the Feature Objects menu to define individual polygon attributes that constitute the table. Information on assigning

parameters to land use and soil type polygon coverages is given on page 3-14. The file formats used for the different mapping parameters are given in section 20-28.

Composite CN Table

Because land use tables with corresponding CNs vary from text to text and agency to agency, *WMS* supports a user-definable method for relating land use to CNs. This is done through a simple table file that is imported prior to computing the composite CN. The file scsland.tbl is an example of such a file and was created from a table given in the Handbook of Hydrology. This file can be edited to supply your own values or a new table with the same format can be created. The format of the table is given in the file formats chapter, section 20.17.

Runoff Coefficient Table

The runoff coefficient table must be defined to relate soil type IDs to runoff coefficient values. A very simplified file named soiltype.tbl is provided with the distribution of *WMS*. This file can be edited to supply your own values or a new table with the same format can be created. The format of the table is given in the file formats chapter, section 20.17.

15.3 Computing Travel Times (Lag and Time Of Concentration)

Lag time and time of concentration are variables often used when computing surface runoff using unit hydrograph methods available in the hydrologic models supported in *WMS*. These variables indicate the response time at the outlet of a watershed for a rainfall event, and are primarily a function of the geometry of the watershed. WMS provides two powerful methods of computing travel times for lag time and time of concentration from the geometric data you are using for basin delineation and parameter estimation.

The first method is to use one of several empirical equations (or user-defined) based primarily on the basin data computed by *WMS* when using a DEM (see section 5.5.6)or TIN (see section 7.7.10) for basin delineation. Many different equations have been developed for different watersheds, and most of these equations are a function of the geometric parameters computed from digital terrain models (see section 15.3.1).

The second method allows you to create a time computation coverage in the map module and then define the "representative" flow path(s) within each basin using arcs, that are used to determine lag or time of concentration. A travel time equation can then be assigned to each arc (length and slope are automatically determined from the arc when a DEM or TIN is present) and the sum of the arc travel times within a basin used for time of concentration or lag

time. Pre-defined equations such as are used by the FHWA or in TR-55 can be selected or user-defined equations developed (see section 15.3.2).

There is not an advantage of one method over the other. Each allows a certain amount of customization and the ability to generate a summary report in a text file or by copying to the clipboard so that these critical input data can be well documented. In general, if time of travel can be determined from a single empirical equation then computing using the basin data will be more convenient whereas if the time of concentration or lag time is determined by combining the time of travel across one or more flow path segments, (overland flow, shallow concentrated flow, channel flow, etc.) then the map data method will likely work best.

Each of the hydrologic models supported by *WMS* that require a lag time, time of concentration, or channel travel time allow you to pick either of the two methods. Buttons adjacent to the input fields allow access to the different methods and the computed result is used to define the input value for the model you are working on (i.e. TR-55 time of concentration, TR-20 lag time, etc.). You may also compute travel times for selected basins or outlets using the *Compute Travel* Time command from the *Calculators* menu. This dialog is shown in Figure 15-2, and will allow you to choose between the two methods for a selected basin (only the map module method is available for a selected outlet). The computed travel time can then be assigned to the relevant input parameter for the selected hydrologic model (the hydrologic model corresponds to the current default model and can be changed using the drop-down combo box).

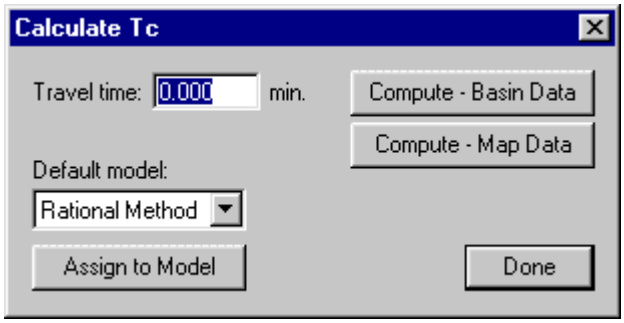


Figure 15-2 Compute Travel Time Dialog.

15.3.1 Computing Travel Times from Basin Data

WMS computes many geometric parameters when using the *Compute Basin Data* command for either a TIN or DEM. These parameters form the basis of the empirical equations used to compute lag time and time of concentration. The *Basin Time Computation* dialog (shown in Figure 15-3) allows you to select from a series of pre-defined equations (Dodson & Associates, 1992), or create your own equation using any of computed basin parameters to compute time of concentration or lag time. If the equation is a function of variable not

computed by *WMS* then you will have to enter the value in this dialog before a travel time can be computed. The process of computing travel times for a selected basin from computed basin data involves the following two steps:

1. Select an equation (or create a new one).

2. Follow the instructions provided in the *Instructions / Results* text window until all of the necessary data are correctly defined (you may also edit any of the variables computed by *WMS*).

The following sections detail these two steps as well as document the basin geometric parameters computed by WMS and the empirical equations that are available to choose from.

Assigning an Equation to a Basin

The *Basin Time Computation* dialog (Figure 15-3) allows you to select from one of several pre-defined equations. The *Equation* drop-down box lists the available methods for computing lag time or time concentration and is controlled by setting the *computation type* combo box to the desired setting. Once you have selected an equation the *Instructions / Results* window will identify any variables with zero values and instruct you to define values for them. Variables are assigned values by selecting the variable from the *Variables* text window and then assigning a value to it in the edit box to the right. Once all variables are defined the *Instructions / Results* window will display the computed lag time or time of concentration.

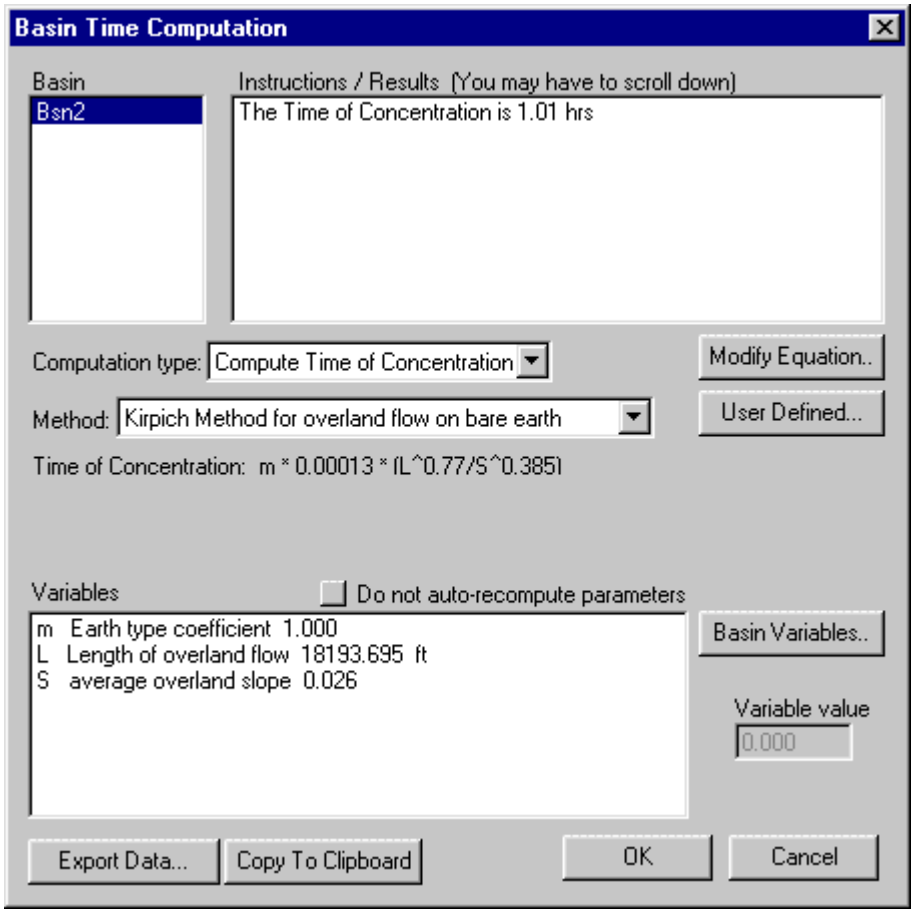


Figure 15-3 Basin Time Computation Dialog.

Auto-recomputing

By default once an equation is specified for a basin, the lag time and time of concentration will be computed automatically each time that basin data are computed, or when the curve number changes. If you do not wish to have the equation updated when basin data changes then you should turn on the *Do not Auto-recompute parameters* check box for the selected basin.

Editing Basin Variables

The *Basin Variables…* button will let you view/edit any of the basin variables that are computed by *WMS*. While it is unnecessary to edit these variables, you may find some cases where you want to over-ride what *WMS* has computed and use a value you have derived through a separate analysis.

Exporting Results for Report Creation

The *Export Data…* and *Copy To Clipboard* buttons are used to create a text report that summarizes the equation, variables, and computed time of concentration or lag time for the basin. Exporting the data will create a text file and allows you to either append to an existing file (so that a single report

for multiple basins can be created) or create a new file. Copying to the clipboard places the report text on the Windows clipboard so that it is available for pasting into other documents.

Customizing Equations

Any one of the equations can be edited (in other words you can pick an equation and then modify it slightly to meet your needs) or your own equation created using the *Modify Equation…* or *User Defined…* buttons. Both of these buttons bring up the Modify Equation dialog shown in Figure 15-4.

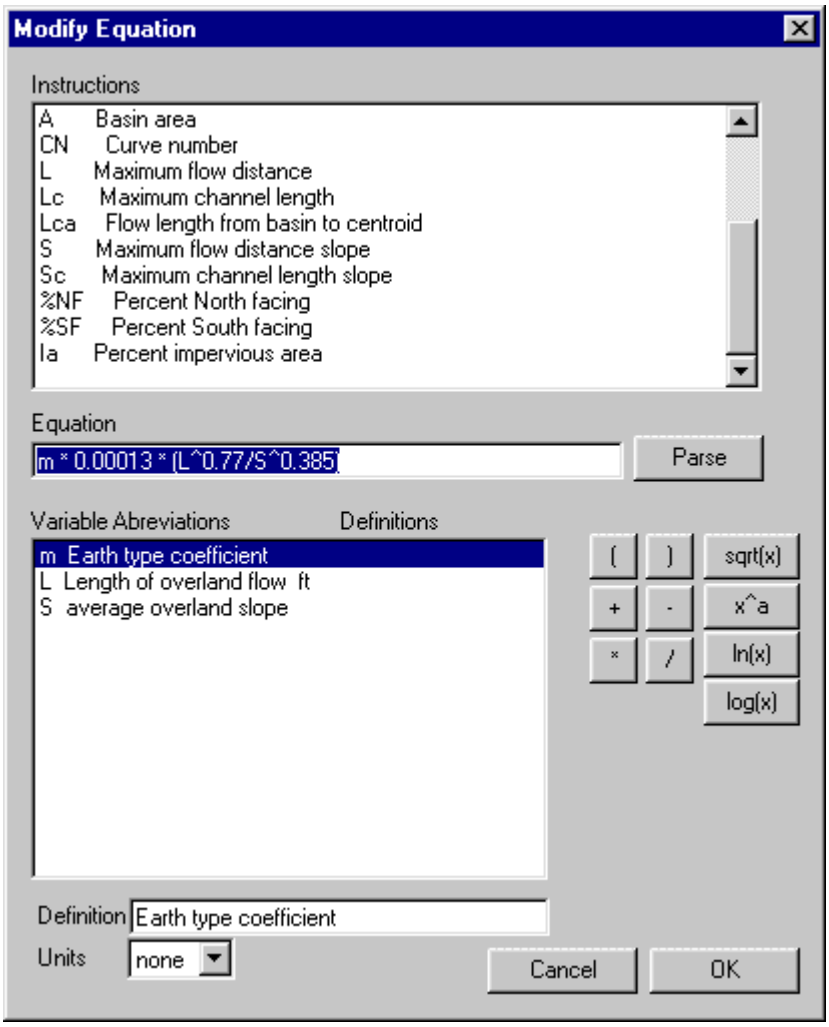


Figure 15-4 Modify Equation Dialog.

If you enter this dialog using the *Modify Equation…* button the currently selected equation will appear in the *Equation* edit box. If the current equation has more than one equation (some equations have separate equations for lag time/time of concentration and peaking or storage coefficients) then you will be asked to choose which equation you want to modify. If you use the *User*

Defined… button to enter this dialog then the *Equation* edit box will be blank. You can type a new or modify an existing equation from the keyboard or by using the buttons for add, subtract, multiply, divide, etc. You may also enter variable names corresponding to the basin data computed by *WMS*, or enter variables that you wish to define yourself. The variable definition is displayed and edited in the *Definition* edit box and units associated with the variable can be set using the *Units* drop-down combo box.

When you have finished typing the equation you should select the *Parse* button. Doing so will result in any variables being identified and displayed in the *Variables* text window along with their definitions. The rules of precedence are typical for mathematical computations and are as follows:

1. Parenthetical statements
2. Functions like ln, log, etc.
3. The power function (^)
4. Multiply and divide
5. Add and subtract
6. Left to right

Repeated Use of a User Defined Equation

If your local hydrology manual requires the use of an equation not currently supported by WMS and you must repeatedly use this equation for projects then you can set up the equation in a text file in the same directory as WMS and it will be read in and set up as the User Defined equation automatically. To do this you must create a file named "wmstc.equ" and define one or more equation groups (a group may be used because sometimes your equation to compute a travel time may be a function of another equation). The following is an example wmstc.equ file.

```
EQGROUP Myeq1
EQ 1.49 / n * R * Sc DEF Manning’s equation UNITS laghour
VAR n DEF Manning’s roughness coefficient VAL .002 UNITS none WHICH -1
VAR R DEF Hydraulic Radius VAL 0.0 UNITS ft WHICH 2
EQ XA / Pw DEF Hydraulic Radius UNITS ft
VAR XA DEF Cross section area VAL 0.0 UNITS sqft WHICH -1
VAR Pw DEF Wetted Perimeter VAL 0.0 UNITS ft WHICH -1
EQGROUP Myequ2
EQ Lc / V DEF Time of travel UNITS lagmin
VAR V DEF Velocity VAL 2.5 UNITS none WHICH -1
```

The following describes the keywords used in the file as well as the possible values that can be entered following each keyword.

Each equation group begins with the EQGROUP card. The rest of the line is interpreted as a character string and used as the name of the equation group in the drop-down combo box allowing you to select an equation.

Each equation is defined using the following three cards with their associated inputs:

EQ - The equation

DEF - A character string definition for the equation

UNITS - Units of the computed result. Possible values for units include: laghour, lagmin, tchour, tcmin, clarkr, tp. The first four identify whether the equation is intended for use in computing lag time or time of concentration (a conversion to the other is done using equation 15.1) and whether the result is in minutes or hours. The last three are used if the equation is used to compute the Clark storage coefficient or one of the peaking parameters used in HEC-1.

If one of the WMS recognized variables are used (A, Lca, etc.) then you do not need to include a VAR record as it will automatically map the WMS variable for use in the equation. For variables not recognized as something WMS computes you should include a VAR line that uses the following keywords:

VAR - Name of the variable

DEF - Variable definition or description

VAL - The initial default value (should be 0.0 if this will be entered separately for each basin)

UNITS - The units of the variable. The following list of keywords are recognized for units. If you use a variable with units not in this list you should enter "none" and then make sure that the equation is dimensionally consistent with the units defined for the equation on the EQ line.

laghour
tcmin
tchour
lagmin
mi mile
m meter
ft feet
km kilometer
hr hour
sec second
none none
acre acre
sqkm square kilometer
sqmi square mile
sqft square feet
sqm square meter
in inch
mm millimeter

hect hectare

WHICH - If a variable represents another equation that is part of the equation group then you should follow the WHICH keyword with the equation number. If it does not represent another equation then you should follow it with a -1. See the example above where the hydraulic radius variable R represents the second equation in the group (XA/Pw).

You may define as many equations as you want in the file and each time WMS is started the equations will be read and become members in the list of equations you can select from when defining travel times from basin data.

Computed Basin and Stream Variables

A complete list of variables computed for drainage basins that are available for use in defining equations, along with the acronyms used in *WMS* is given below:

A The area of the basin in the units specified prior to computing basin parameters.

BS The average basin slope, or average slope of the triangles comprising this basin. A triangle's slope is computed as the change in elevation divided by the change in XY or plan distance.

AOFD The average overland flow distance within the basin. This is computed by averaging the overland distance traveled from the centroid of each triangle to the nearest stream.

%NF The percentage of the basin whose aspect is directed North where North is defined as the positive Y direction.

%SF The percentage of the basin whose aspect is directed South where South is defined as the negative Y direction.

L Basin length.

P Perimeter of the basin.

Shape The shape factor of the basin, or the length divided by the width.

Sin Sinuosity factor of the stream in the basin. Defined by dividing the maximum stream length in the basin by the length.

AVEL The mean basin elevation.

MFD The maximum flow distance within a basin including both overland and channel flow.

MFDS The slope of the MFD.

CTOMFD The distance from the centroid of the basin to the nearest point associated with the MFD.

CSD The distance from the centroid of the basin to a point in the stream which is a part of the MFD. The CSD differs from the CTOMFD in that it is only concerned with the channel or stream flow portion of the MFD, whereas the CTOMFD also incorporates the portion of the MFD which is overland flow.

CSS The slope of the CSD.

MSL The maximum stream length within the basin. This is computed by determining the maximum distance traveled when "flowing" down from the top of streams in a basin and where the streams exit the basin.

MSS The slope of MSL.

In addition to the basin attributes defined above the following stream attributes are computed:

L Stream lengths for each segment.

SS Stream slopes for each segment.

Stream attribute labels cannot be moved.

Empirical Lag Time and Time of Concentration Equations

Dodson (Dodson & Associates, 1992) compiled several equations from hydrologic publications. *WMS* has implemented many of these equations and allows you to choose from the ones listed below to automatically compute lag times/times of concentration. Because most of the equations were developed for specific watersheds (i.e. size, land cover etc.) you should consider the assumptions made about a given equation, and try to identify one that used watershed conditions similar to the one you are studying. The following is a list of the equations available in *WMS*. The SCS also found that for many cases the lag time could be related to the concentration time by the following equation:

$$T_{LAG} = 0.6 * T_c \quad \text{15.1}$$

This relationship is always used by *WMS* to determine lag time when a method for computing time of concentration is chosen, or to compute time of concentration when a method for lag time is chosen. Methods used for lag

time begin with "T_{LAG} =" whereas methods used to compute time of concentration begin with "T_c =".

Customized Lag Time Equation

Almost all of the lag time equations are of the form:

$$T_{LAG} = C_t * \left(\frac{(L * L_{ca})}{\sqrt{S}} \right)^m \quad \text{..... 15.2}$$

where:

C_t coefficient accounting for differences in watershed slope and storage

L the maximum flow length of the watershed along the main channel from the point of reference to the upstream boundary of the watershed, in miles.

L_{ca} the distance along the main channel from the point of reference to a point opposite the centroid, in miles.

S Slope of the maximum flow distance path in ft/mile.

m lag exponent

Therefore, if the equation that your state, county, etc. uses to compute lag time is not available, it can often be set up using this equation by entering your own coefficient C_t and exponent m.

Tulsa District Lag Time Equation

The Tulsa District of the US Army Corps of Engineers has developed the following family of equations for computing Snyders lag time:

$$T_{LAG} = C_t * \left(\frac{L * L_{ca}}{\sqrt{S}} \right)^{.39} \quad \text{..... 15.3}$$

where:

C_t 1.42 for natural watersheds in rural areas of central and northeastern Oklahoma.

C_t 0.92 for the same type areas that are 50% urbanized.

C_t 0.59 for the same type areas that are 100% urbanized.

L watershed maximum flow distance length in miles.

S Slope of the maximum flow distance path in ft/mile.

L_{ca} length to centroid.

The range of watershed characteristics for which these equations apply include:

- Sizes ranged from .5 to just over 500 square miles.
- Slopes ranged from 4 to 90 feet per mile.
- Lengths ranged from 1 to 80 miles.
- Length to centroid ranged from 1 to 60 miles.

In addition to developing an equation for lag time, the Tulsa district developed the following relationship the peak flow rate which can be used in equation 15.5 to solve for Snyder's peaking coefficient.

$$q_p = 380 * T_{LAG}^{-.92} \quad \text{.....15.4}$$

$$C_p = \frac{q_p * T_{LAG}}{640} \quad \text{.....15.5}$$

Denver Lag Time Equation

Equation 15.6 was developed by the Denver Area Urban Drainage and Flood Control District (Wright-McLaughlin Engineers, 1975)

$$T_{LAG} = C_t * \left(\frac{L * L_{ca}}{\sqrt{S}} \right)^{.48} \quad \text{.....15.6}$$

where:

C_t Time to peak coefficient.

L length along the stream from the study point to the upstream limits of the basin in miles.

L_{ca} length along the stream from the study point to a point along the stream adjacent to the centroid of the basin in miles.

S weighted average slope of the basin from the study point to the upstream limits of the basin in feet per foot.

The percent impervious (I_a) must already be defined in one of the Loss methods used for HEC-1.

This equation was developed for small urban watersheds (less than 5 square miles) with mild slopes. The peaking coefficient can be computed from the percent impervious (I_a) using the following equations:

$$C_t = -0.00371 I_a + 0.163 \qquad 0 \le I_a \le 10 \quad \text{..... 15.7}$$

$$C_t = 0.000023 {I_a}^2 - 0.002241 I_a + 0.146 \qquad 10 \le I_a < 40 \quad \text{..... 15.8}$$

$$C_t = 0.0000033 {I_a}^2 - 0.000801 I_a + 0.12 \qquad 40 < I_a \le 100 \quad \text{..... 15.9}$$

The Denver method used a peaking parameter P and the relationships below to compute the peaking coefficient C_p.

$$P = 0.002450 {I_a}^2 - 0.0120 I_a + 2.16 \qquad 0 \le I_a \le 40 \quad \text{..... 15.10}$$

$$P = -0.00091 {I_a}^2 + 0.228 I_a - 2.06 \qquad 40 \le I_a \le 100 \quad \text{..... 15.11}$$

$$C_p = P * C_t * A^{0.15} \quad \text{..... 15.12}$$

where:

C_t coefficient as defined by equations 15.7 to 15.9.

P peaking parameter.

A basin area in square miles.

Espey Lag Time Equation

Espey's equations for Snyder's parameters were developed for a series of small watersheds in Texas, Oklahoma, and New Mexico. Rather than defining the lag time, Espey (1966) used the time to rise. The difference is that the lag time is the time from the centroid of rainfall to the peak of the hydrograph, whereas the time to rise is the time from the beginning of effective rainfall to the peak of the hydrograph. The lag time can be computed by subtracting one-half the computation time interval from the time rise. Equations to compute T_r and C_p are given below:

$$T_r = 2.65 {L_f}^{0.12} {S_f}^{-0.52} \qquad \text{Rural Areas ..... 15.13}$$

$$T_r = 20.8 U {L_f}^{0.29} {S_f}^{-0.11} {I_a}^{-0.61} \qquad \text{Urban Areas..... 15.14}$$

where:

T_r time from the beginning of effective rainfall to the peak of the unit hydrograph.

L_f stream length in feet.

S_f stream slope in feet per foot

I_a percent impervious cover

U Urbanization factor equal to .6 for extensive urbanization, .8 for some storm sewers, and 1.0 for natural conditions.

Typical conditions for typical rural watersheds include:

- L_f from 3250 feet to 25,300 feet.
- S_f from 0.008 ft/ft to 0.015 ft/ft.
- T_r from 30 to 150 minutes.
- Areas from .1 sq. miles to 7 sq. miles.

Typical conditions for typical urban watersheds include:

- L_f from 200 feet to 54,800 feet.
- S_f from 0.0064 ft/ft to 0.104 ft/ft
- I_a from 25 to 40 percent.
- T_r from 30 to 720 minutes
- Areas from .0125 sq. miles to 92 square miles.

Espey developed the following equations to compute Snyders peaking coefficient.

$$q_p = 1700 A^{-0.12} T_r^{-0.30}$$ Rural Areas 15.15

$$q_p = 19300 A^{-0.09} T_r^{-0.94}$$ Urban Areas 15.16

Once qp is computed the peaking coefficient can be determined using the following relationships:

$$T_{LAG} = T_r - \Delta t / 2$$ 15.17

$$C_p = \frac{q_p * T_{LAG}}{640}$$ 15.18

where Δt is the computational time interval as define in the *HEC-1 Job Control* dialog (by default it is 15 minutes).

Riverside County Lag Time Equations

The Riverside County Flood Control and Water Conservancy District developed three different lag equations corresponding to mountainous, foothill, and valley areas near Riverside County, California (Anonymous, 1963).

$$T_{LAG} = 1.20\left(\frac{L * L_{ca}}{\sqrt{S}}\right)^{0.38} \quad \text{(Mountainous)} \quad \text{15.19}$$

$$T_{LAG} = 0.72\left(\frac{L * L_{ca}}{\sqrt{S}}\right)^{0.38} \quad \text{(Foothills)} \quad \text{15.20}$$

$$T_{LAG} = 0.38\left(\frac{L * L_{ca}}{\sqrt{S}}\right)^{0.38} \quad \text{(Valleys)} \quad \text{15.21}$$

where:

T_{LAG} the lag time in hours.

L maximum flow length in miles.

L_{ca} length to the centroid in miles.

S weighted slope along the maximum flow path length in ft/mile.

The typical characteristics of watersheds for which the Riverside County equations were used include the following:

- Areas from 2 to 650 square miles.

Eagleson Lag Time Equation

Eagleson's method (1962) for computing lag time in completely storm-sewered watersheds is given in equation 15.22.

$$T_{LAG} = 0.32\left(\frac{L * L_{ca}}{\sqrt{S}}\right)^{0.39} \quad \text{15.22}$$

where:

T_{LAG} the lag time in hours.

L maximum flow length in miles.

L_{ca} the length to the centroid in miles

S weighted average slope of the maximum flow path in ft/mile.

The typical characteristics of watersheds for which the Eagleson method was applied include the following:

- Areas from .22 to 7.5 square miles.
- L from 1 to 7 miles.
- L_{ca} from .3 to 3 miles.
- S from 6 to 20 ft/mile.
- Impervious cover from 33 to 83 percent.

Putnam Lag Time Equation

Putnam (1972) developed a lag equation for watersheds around the Wichita, Kansas area as follows:

$$T_{LAG} = 0.49\sqrt{\frac{L}{\sqrt{S}}}I_a^{-0.57} \quad \text{.....15.23}$$

where:

T_{LAG} lag time in hours.

L maximum flow length in miles.

S weighted slope along the maximum flow path in ft/mile.

I_a Impervious cover as a fraction.

This equation was used for watersheds ranging in size from .3 to 150 sq. miles, impervious covers less than .3 and a ratio of $L/\sqrt{S}$ between 1.0 and 9.0.

Taylor and Schwartz Lag Time Equation

Taylor and Schwartz (1952) developed an equation for estimating Snyder unit hydrograph parameters that was used for 20 different watersheds in the northeastern region of the U.S. Their equations are as follows:

$$C_t = \frac{0.6}{\sqrt{S}} \quad \text{.....15.24}$$

$$T_{LAG} = C_t * \left(L * L_{ca}\right)^{0.3} \quad \text{.....15.25}$$

where:

C_t coefficient of watershed topography based on watershed slope.

S weighted slope of maximum flow path in ft/ft.

T_{LAG} the watershed lag time in hours.

L maximum flow length in miles.

L_{ca} length to the centroid in miles.

Colorado State Lag Time Equation

An equation used to compute lag time for watersheds in the Denver Colorado area was developed at Colorado State University. This equation was primarily used for watersheds in which there was some amount of developed land. It is not valid for watersheds with less than 10 percent impervious area (I_a). The equation uses a runoff coefficient which represents variations in topography. This coefficient can also be used to compute the peaking factor using the relationship shown in Figure 15-5.

$$C_t = \frac{7.81}{I_a^{0.78}} \quad \text{..... 15.25}$$

$$T_{LAG} = C_t * \left(L * L_{ca}\right)^{0.3} \quad \text{..... 15.26}$$

where:

C_t coefficient of watershed topography based on impervious area.

I_a Percentage of impervious area in the watershed (must be defined in the HEC-1 Loss methods).

T_{LAG} the watershed lag time in hours.

L maximum flow path length in miles.

L_{ca} length to the centroid in miles.

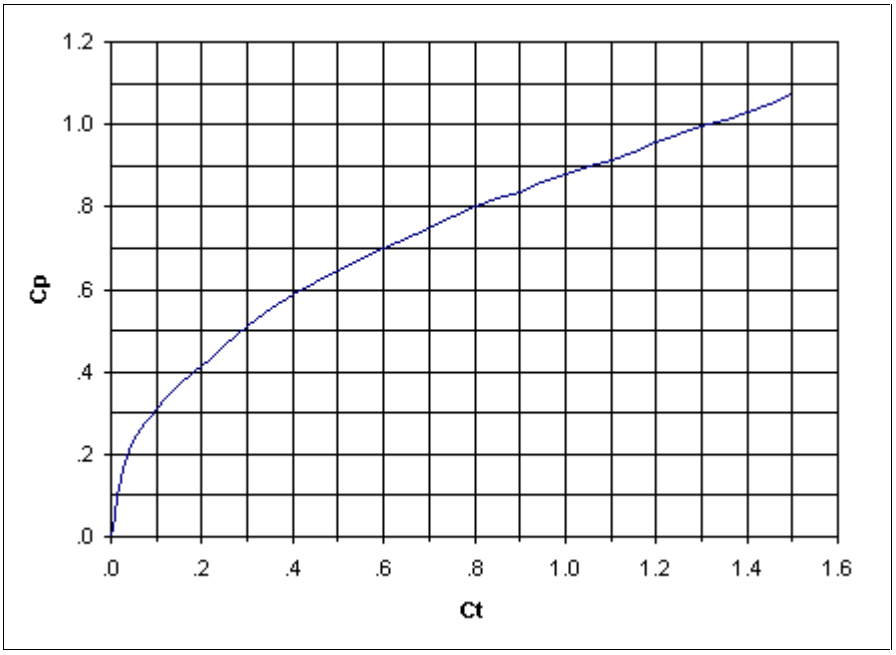


Figure 15-5 Graph for Determining the Peaking Coefficient C_p from C_t.

SCS Lag Time Equation

Perhaps the most commonly used equation for lag time is the SCS equation (1972) given in equation 15.27. This equation may be used when computing the unit hydrograph using Snyder's method and any of the preceding equations for lag time may also be used when computing the unit hydrograph using the SCS method. Also remember that the SCS used the relationship defined in equation 15.1 to compute lag time (and the other way around) from any of the time of concentration equations which follow.

$$T_{LAG} = L^{0.8} \frac{(S+1)^{0.7}}{1900\sqrt{Y}} \quad \text{.....15.27}$$

where:

T_{LAG} Lag time in hours.

L Hydraulic length of watershed in feet.

S Maximum retention in the watershed in inches as defined by equation 15.26.

$$S = \frac{1000}{CN} - 10 \quad \text{.....15.28}$$

Y Watershed slope in percent.

CN SCS curve number for the watershed as defined by the loss method.

Kerby Time of Concentration Equation

Kerby (1959) developed an equation for computing the time of concentration for overland flow distances of less than 500 feet and greater than 300 feet.

$$t_c = \left(\frac{0.67 * n * L_O}{\sqrt{S}} \right)^{0.467} \quad \text{..... 15.29}$$

where:

t_c time of concentration in minutes.

S overland slope in ft/ft.

n roughness coefficient.

L_O length of overland flow in feet.

A table of recommended values for n is given in table 15.1.

Table 15.1. Recommended surface roughness values

Surface Description	n
Smooth, impervious surface	0.02
Smooth, packed bare soil	0.1
Poor grass, cultivated row crops of moderately rough bare soil	0.2
Pasture or average grass	0.4
Deciduous timberland	0.6
Timberland with deep forest litter or dense grass	0.8

Kirpich Time of Concentration Equation

Kirpich's equation (1940)was developed for small, agricultural watersheds. It was derived by examining the required time for the stream to rise from low to maximum stage during a storm. The time of concentration was then assumed equal to that time.

$$t_c = \frac{0.00013 L^{0.77}}{S^{0.385}} \quad \text{..... 15.30}$$

where:

t_c time of concentration in hours.

L length of the overland flow in feet.

S average overland slope in ft/ft.

This equation given in 15.31 was developed for overland flow on bare earth. For overland flow on grassy earth t_c should be multiplied by 2.0. On concrete and asphalt surface it should be multiplied by 0.4. An adjustment is made for watersheds with a CN number less than 80 using the following equation:

$$t_c = t_c * \left(1 + (80 - CN) * .04\right) \quad \text{.....15.31}$$

The CN value must be defined for the given model (HEC-1, TR-20, etc.), otherwise a default CN of 50 is used.

Ramser Time of Concentration Equation

Ramser (1927) developed an equation for computing the time of concentration in well-defined channels. The equation is based on the length and slope of the channel.

$$t_c = 0.008 L_c^{0.77} S_c^{-0.385} \quad \text{.....15.32}$$

where:

tc time of concentration in minutes

Lc length of channel reach in feet.

S_c average channel slope in ft/ft.

For flow in concrete channels t_c should be multiplied by 0.2.

Fort Bend County Equation

The county of Fort Bend Texas (Espey, Huston, & Associates, 1987) used the equation shown in equation 15.33 to compute t_c. In addition to defining an equation for time of concentration to be used in the Clark unit hydrograph method, they also defined a relationship for the Clark watershed storage coefficient that is given by equation 15.34.

$$t_c = 48.64 \frac{\left(\frac{L}{\sqrt{S}}\right)^{0.57}}{S_o^{0.11} 10^I} \log S_o \quad \text{.....15.33}$$

$$R = 128 \frac{\left(\frac{L}{\sqrt{S}}\right)^{0.57}}{S_o^{0.11} 10^I} \log S_o - t_c \quad \text{.....15.34}$$

where:

t_c Clark time of concentration in hours.

R Clark watershed storage coefficient.

L Length of longest flow path within the watershed in miles.

S Average slope along the longest flow path.

S_o Average basin slope.

I Percent impervious as a fraction.

Typical characteristics of the watersheds for which these equations were applied are:

- Area between .13 and 400 square miles.
- Length of longest flow path between .5 and 55 miles.
- Slope of longest flow path from 2 ft/mi. to 33 ft/mi.
- Slope of basin from 3 to 80 ft/mi.
- Impervious area from 0 to 100%.

Others have simply used the simple relationship defined by equation 15.35 to compute the Clark watershed storage coefficient from the time of concentration.

$$R = 2t_c \quad \text{..... 15.35}$$

15.3.2 Computing Travel Times from Map Data

The *Time Computation* coverage (see page 3-18) can be used to create arcs representing flow path segments when computing time of concentration or lag time for a basin or reach. Within a basin the time of concentration or lag time is usually determined by combining the time of travel across one or more flow path segments. Travel time equations are generally functions of the length and slope of the flow path segment as well as surface roughness (i.e. Manning's roughness coefficient), and channel shape and roughness. Since length and slope (providing there is a background TIN or DEM) are easily determined from arcs, the time computation coverage provides a simple and powerful method for computing basin time of concentration and/or lag time. Travel times between consecutive outlet points may also be computed using the same tools.

If you have developed your watershed model from a TIN or DEM you can have WMS automatically create flow path arcs from selected points using the *Node->Flow Arcs* and *Stream->Flow Arcs* command (see page 3-36).

The process of computing a time of concentration involves two primary steps.

1. Compute travel times for individual arc segments.

2. Combine the travel times of all arcs within a basin to compute the time of concentration for the basin, or combine arcs between outlet points to compute the travel time along a stream reach.

The next two sections describe how these two tasks are performed using *WMS*.

Assigning Equations to Time Computation Arcs

The primary attribute for a time computation arc is the equation that will be used to compute travel time for the flow path segment represented by the arc. In addition to the equation the length, slope, and travel time for the arc are also stored. Length is defaulted from the length of the arc and a slope will be determined for the arc if there is an underlying TIN or DEM. However, you can edit either of these values if you want. For example you may decide that the actual flow path is somewhat more sinuous than the arc represents and decide to increase the length. The equation and appropriate variables are defined for an arc using the *Time Computation Arcs Attribute* dialog shown in Figure 15-6.

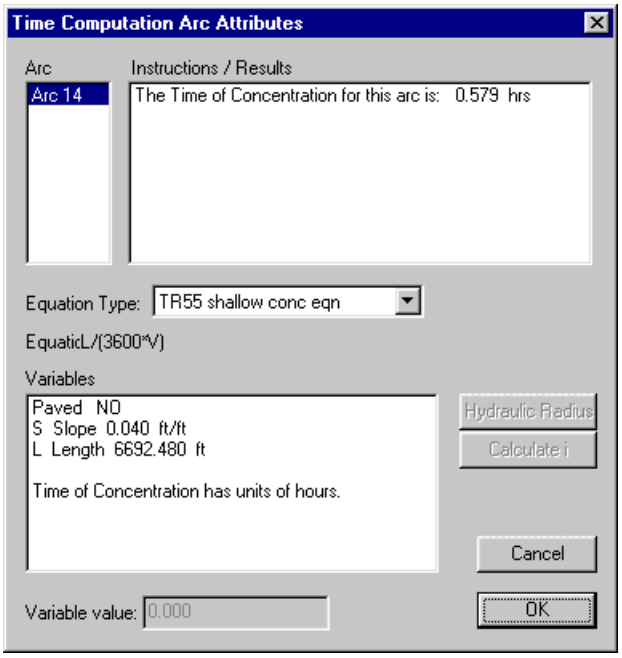


Figure 15-6 Time Computation Arc Attributes Dialog.

This dialog is used to set all values and compute travel times for selected arcs. It can be accessed by selecting an arc and then choosing the *Attributes* command from the *Feature Objects* menu in the map module, by double clicking on the arc when in the map module, or when using the *Travel Time*

Computation dialog (see Figure 15-8). If multiple arcs are selected then the ArcID window displays the ID of all selected arcs and you may edit the equations / variables of any selected arc by choosing it from this window. While the selected arc is highlighted in the *Graphics Window*, you may find it useful to toggle on the display of arc ID's from the *Feature Object Display Options* dialog (see section 3.2.7).

When using the *Time Computation Arc Attributes* dialog the *Instructions / Results* window will let you know which variables need to be entered before a time computation can be made, and when all variables are defined it will display the computed travel time of the selected arc.

Editing Equation Variables

Equation variables from the currently selected arc are displayed in the variables window of the *Time Computation Attributes* dialog. Variables such as length and slope will generally have defaulted values, however other variables such as Manning's roughness coefficient will need to be entered before a travel time for the arc can be computed. Variables are edited by selecting the variable you want to modify from the text window and then setting the value in the adjacent edit field. The *Instructions/Results* window will let you know when variables have not been defined and the travel time for the arc once all variables have reasonable values.

Equation Types

The *TR-55* equations for travel time are one set of commonly used equations to compute time of concentration. Others, including those used by the Federal Highways Administration are variations of the same type of equations. These sets of equations form a library of predefined equations in *WMS*. However, you can also enter your own equation, or modify one of the existing equations.

Equations consist of a type and sub-type (user-defined equations do not require a sub-type). Both are specified using the drop-down combo boxes in the attribute dialog.

TR-55

The *TR-55* equations represent three different flow conditions: sheet flow, shallow concentrated flow, and open channel flow. To select a *TR-55* equation set the type to *TR-55* and the sub-type to one of the three flow conditions.

Sheet Flow

Sheet flow generally occurs in the headwater of streams. The following equation is used to describe sheet flow:

$$T_t = \frac{0.007(nL)^{0.8}}{(P_2)^{0.5} s^{0.4}} \quad \text{..... 15.36}$$

where:

T_t – travel time (hr).
n = Manning's roughness coefficient (see Table 3-1 of the *TR-55*)
L – flow path length in feet
P_2 – 2year, 24 hour rainfall (in)
S – slope of the hydraulic grade line (ground slope) in ft/ft.

2 year, 24 hour rainfall values can be determined from the map in appendix B-4 of the *TR-55* reference manual for eastern US locations and from the NOAA Atlas 2 maps for the western US locations.

Generally the sheet flow equation should not be used for lengths greater than 300 feet.

Shallow Concentrated Flow

After 300 feet sheet flow usually turns to shallow concentrated flow. The following equation is used to compute travel time for shallow concentrated flow:

$$T_t = \frac{L}{3600V} \quad \text{.....15.37}$$

where:

T_t - Travel time (hr).
L – Flow length in feet.
V – Average velocity (f/s).
3600 – Conversion factor from seconds to hours.

Average velocities are determined from the slope of the flow path segment and are slightly different depending on whether the surface is paved or unpaved. Equations from Appendix F of the *TR-55* reference are used in *WMS* to convert the slope to an average velocity. Pavement is a variable that must be defined if this equation type is specified for the arc.

Open Channel Flow

Travel time for open channel flow segments is computed using equation 15.37 here the average velocity is determined using Manning's equation for open channel flow:

$$V = \frac{1.49}{n} R^{2/3} S^{1/2} \quad \text{.....15.38}$$

where:

V - Average channel velocity.
n – Manning's roughness coefficient for open channel flow.
R - hydraulic radius (ft) and is equal to A/P_w.
A – Cross-sectional area of flow (sq. feet).

P_w – Wetted perimeter (ft).
S – slope of the hydraulic grade line (channel slope).

Slope values are determined from the arc when a TIN or DEM is present. The channel calculator (see section 15.4) can be used to determine the hydraulic radius. In this case you will have to assume an approximate depth of flow or flow rate in order for the channel calculator to be able to compute the appropriate hydraulic radius.

Federal Highways

The FHWA equations are taken from the Federal Highways Drainage manual. Similar to the *TR-55* equations defined above travel time within a basin is determined from three different flow conditions: sheet flow, shallow concentrated flow, and open channel flow. However, the equations for the three different conditions are slightly different.

Sheet Flow

Sheet flow generally occurs for the first 300 feet at the headwater of streams. The following equation is used to describe sheet flow:

$$T_t = \left(\frac{K}{i^{0.4}}\right)\left(\frac{nL}{\sqrt{s}}\right)^{0.6} \quad \text{..... 15.39}$$

where:

K – emprical coefficient equal to .933 for English units and 6.943 for Metric.
i – rainfall intensity (in/hr).
n – Mannings roughness coefficient for overland flow. Suggested values are given in Table 3-2 of the FHWA HEC 22 manual and are repeated in the table below.
L – length of the overland flow segment (ft).
S – ground slope of the flow segment (ft/ft).

Manning's Roughness for overland sheet flow	
Surface Description	**n**
Smooth asphalt	0.011
Smooth concrete	0.012
Ordinary concrete lining	0.013
Good wood	0.014
Brick with cement mortar	0.014
Vitrified clay	0.015
Cast iron	0.015
Corrugated metal pipe	0.024
Cement rubble surface	0.024
Fallow (no residue)	0.05
Cultivated soils	
Residue cover ≤ 20%	0.06
Residue cover > 20%	0.17
Range (natural)	0.13
Grass	

Short prairie grass	0.15
Dense grasses	0.24
Bermuda grass	0.41
Woods	
Light underbrush	0.40
Dense underbrush	0.80

The rainfall intensity is actually a function of the travel time for the flow segment. In order to iteratively solve for the travel time you must define an IDF curve (function) to be used in conjunction with the equation. IDF curves are defined in *WMS* using Hydro 35, NOAA Atlas 2, or user defined rainfall intensities for specific durations. The *IDF Curves* dialog used as part of the rational method is used to set up equations relating i to T_t. See section 13.3.3 for more information on developing IDF curves.

Shallow Concentrated Flow

After 300 feet, sheet flow usually turns to shallow concentrated flow. The following equation, based entirely on the length and slope of the arc, is used to compute the travel time for the shallow concentrated segment of flow:

$$T_t = \frac{L}{60k\sqrt{S}} \quad \text{.....15.40}$$

where:

L – Length of flow segment.
k – intercept coefficient (values are given in Table 3-3 of the FHWA HEC 22 manual and are repeated in the table below).
S – slope of the ground surface as a percent.

Intercept coefficients for velocity vs slope relationships of equation 16.5	
Land Cover / Flow Regime	**K**
Forest with heavy ground litter; hay meadow	0.076
Trash fallow or minimum tillage cultivation; contour or strip cropped; woodland	0.152
Short grass pasture	0.213
Cultivated straight row	0.274
Nearly bare and untilled; alluvial fans in western mountain regions	0.305
Grassed waterways	0.457
Unpaved	0.491
Paved area; small upland gullies	0.619

Open Channel Flow

Travel time for open channel flow segments is computed using the following form of Manning's equation for open channel flow:

$$T_t = \frac{Ln}{60KR^{2/3}\sqrt{S}} \quad \text{.....15.41}$$

where:

L – open channel flow length.
n – Manning's roughness coefficient for channel flow. Suggested values are given in Table 3-4 of the FHWA HEC 22 manual and are repeated in the table below.
K – empirical coefficient equal to 1.49 for English units and 1.0 for Metric.
R – hydraulic radius (ft).
S – channel slope (ft/ft).

Values of Manning's coefficient for channels and pipes	
Conduit Material	**n**
Closed conduits	
Asbestos-cement pipe	0.011-0.015
Brick	0.013-0.017
Cast iron pipe	
Cement lined & seal coated	0.011-0.015
Concrete (monolithic)	0.012-0.014
Concrete pipe	0.011-0.015
Corrugated-metal pipe (.5-2.5 inch corrugations)	
Plain	0.022-0.026
Paved invert	0.018-0.022
Spun asphalt lined	0.011-0.015
Plastic pipe (smooth)	0.011-0.015
Vitrified clay	
Pipes	0.011-0.015
Liner plates	0.013-0.017
Open channels	
Lined channels	
Asphalt	0.013-0.017
Brick	0.012-0.018
Concrete	0.011-0.020
Rubble or riprap	0.020-0.035
Vegetal	0.030-0.040
Excavated or dredged	
Earth, straight and uniform	0.020-0.030
Earth, winding, fairly uniform	0.025-0.040
Rock	0.030-0.045
Unmaintained	0.050-0.14
Natural channels (minor streams, top width at flood stage < 100 feet)	
Fairly regular section	0.03-0.07
Irregular section with pools	0.04-0.10

The hydraulic radius may be computed using the channel calculator (see section 15.4). In this case you will have to assume an approximate depth of flow or flow rate in order for the channel calculator to be able to compute the appropriate hydraulic radius.

Maricopa County

The Maricopa County, Arizona equation for computing time of concentration has also been included in *WMS* as a pre-defined equation.

$$tc = 11.4\sqrt{L}K_b^{0.52}S^{-0.31}i^{-0.38} \quad \text{.....15.42}$$

where:

L – length of the flow path (ft).

K_b – representative watershed resistance coefficients. Values are computed using the equation Kb = m log(A) + b where m and b are defined in Table 5.1 of the Maricopa County drainage manual (shown below) and A is the drainage area in acres.

S – ground slope of the flow segment.

i – rainfall intensity (in/hr).

Parameters for Estimating Kb = mlog(A)+b in the Maricopa County Tc Equation

Type	Description	Typical Applications	Parameters	
			m	B
A	Minimal roughness: Relatively smooth and/or well graded and uniform land surfaces. Surface runoff is sheet flow.	Commercial/Industrial areas, Residential area, Parks and golf courses	-0.00625	0.04
B	Moderately low roughness: Land surfaces have irregularly spaced roughness elements that protrude from the surface but the overall character of the surface is relatively uniform. Surface runoff is predominately sheet flow around the roughness elements.	Agricultural fields Pastures Desert rangelands Undeveloped urban land	-0.01375	0.08
C	Moderately high roughness: Land surfaces that have signigicant large to medium sized roughness elements and/or poorly graded land surfaces that cause flow to be diverted around roughness elements. Surface runoff is sheet flow for short distances draining into meandering drainage paths.	Hillslopes Brushy alluvial fans Hilly rangelands Disturbed land, mining, etc. Forests with underbrush	-0.025	0.15
D	Maximum roughness: Rough land surfaces with torturous flow paths. Surface runoff is concentrated in numerous short flow paths that are oblique to the main flow direction	Mountains Some wetlands	-0.030	0.20

Conversion of watershed area between sq. miles and acres may be required in order for this equation to compute the proper time of concentration.

Because the appropriate rainfall intensity value is a function of t_c, you will need to define an IDF curve (equation). The *IDF Curves* dialog used as part of the rational method is used to set up equations relating i to t_c. See section 13.3.3 for more information on developing IDF curves.

User Defined

Because it would be impossible to contain all possible equations used for computing travel times, a user defined equation may be defined for any arc segment. When "user defined" is selected for the arc type the *Modify Equation* button in the *Time Computation Attributes* dialog (Figure 15-6) is active. By selecting this button you can use the *Modify Equation* dialog (see Figure 15-7) to create/modify a suitable equation. User defined equations can be created by typing in the equation edit field using the following rules for precedence:

1. Parts of the equation in parentheses have the highest precedence.

2. Multiplication and division have higher precedence than addition and subtraction.

3. Equations are evaluated from left to right.

Besides typing in the equation you can also select one of the pre-defined equations from the *Sample equations* drop-down list and add it to the equation line as a starting point to create a new equation. You may also use any of the variables that *WMS* can manage as part of the equation (i.e. length, slope, rainfall intensity, etc.). However, you cannot add a new "variable" since there is no way for *WMS* to manage it. If a variable that is not managed by *WMS* is used in your equation you must determine what the appropriate value for the selected arc would be and enter it as a constant in the user defined equation. Variables are added to an equation by either typing the abbreviation, or you may select the variable you want to use and click on the *Add to Equation* button. In a similar fashion mathematical operators can either be typed or the corresponding button selected to add it to the equation.

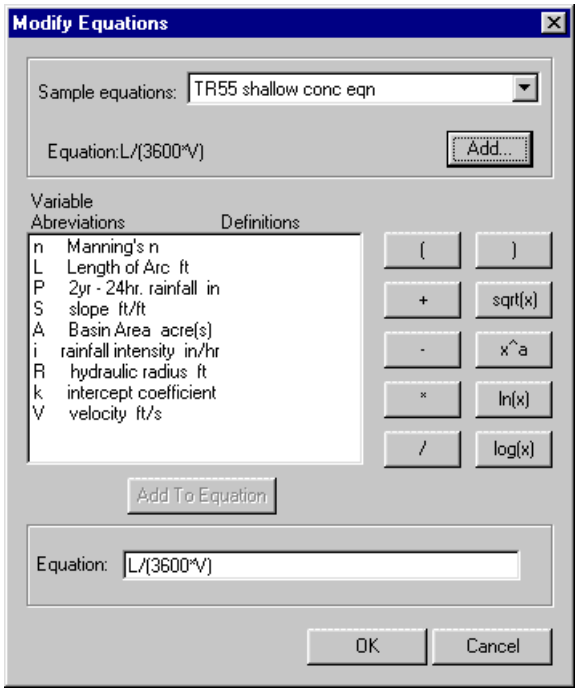


Figure 15-7 Create/Modify Equations Dialog.

Combining Arc Travel Times for Sub-basins or Reaches

After defining equations and variables for individual flow path segments (arcs), the second step in computing a basin time of concentration (or lag time) is to sum the travel times of all arcs within a basin or between outlets. Selecting the basin or outlet and choosing the *Travel Time Computation…* option from the *Calculators* menu in the *Hydrologic Modeling* module allows you to do this. This option is also available directly from within many of the

hydrologic model parameter dialogs where time of concentration, lag time, or routing travel time is needed.

When the *Travel Times Computation* dialog is accessed (see Figure 15-8), you will see the list of time computation arcs that lie within the currently selected basin if a basin is selected or between the selected outlet and the next downstream outlet if an outlet is selected. If you have already defined the equation and necessary variables, the travel time for each arc will be displayed and the total travel time for all arcs will be displayed in the travel time edit field. You can accept the computed value for travel time, or you may edit the value (override computed value) as may be appropriate. If you want to change the equation definitions or variable values for any/all of the arcs you can select the *Edit Arcs* button. This will bring up the *Time Computation Arc Attributes* dialog (see Figure 15-6) which is used for editing travel time equations and variables as described in the previous section.

The *Default Model* button can be selected to determine which set of model parameters is to be assigned the computed time of concentration or travel time.

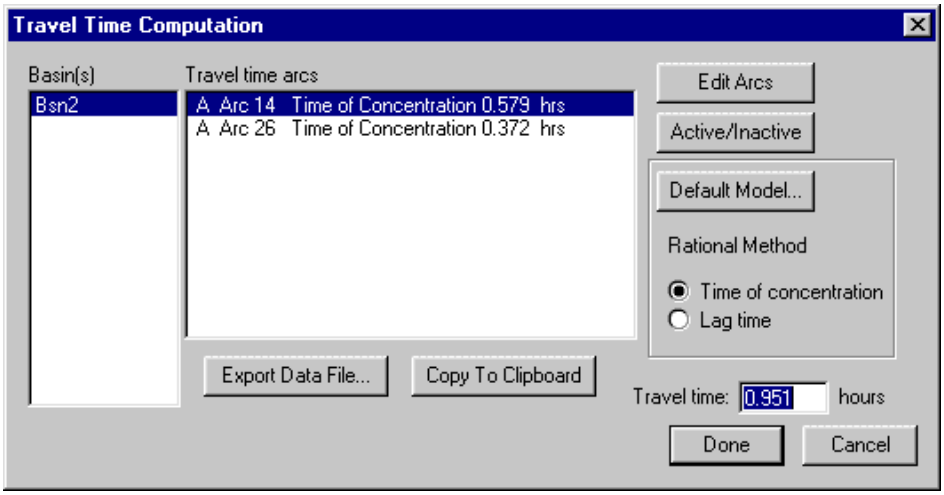


Figure 15-8 Travel Time Computation Dialog.

The *Export Data File…* and *Copy To Clipboard* buttons are used to create a text report that summarizes the equation, variables, and computed time of concentration or lag time for the basin. Exporting the data will create a text file and allows you to either append to an existing file (so that a single report for multiple basins can be created) or create a new file. Copying to the clipboard places the report text on the Windows clipboard so that it is available for pasting into other documents.

15.4 Channels

It is useful to be able to analyze the conveyance and other properties of channels using Manning's equation. The Channels calculator allows for the definition of rectangular, trapezoidal, triangular, circular, and user-defined cross sectional channels. Once channel input geometry is specified, either depth or flow can be computed after supplying a value for the other. If a hydrograph has been computed using one of the supported hydrologic models, the peak flow for the hydrograph will be used as the default flow value if the hydrograph is selected prior to opening the *Channel Calculations* dialog (Figure 15-9). User-defined cross-sections are defined from a cross section coverage and can be interpolated from a background TIN or DEM (see page 3-19 in the coverage type section 3.2.6).

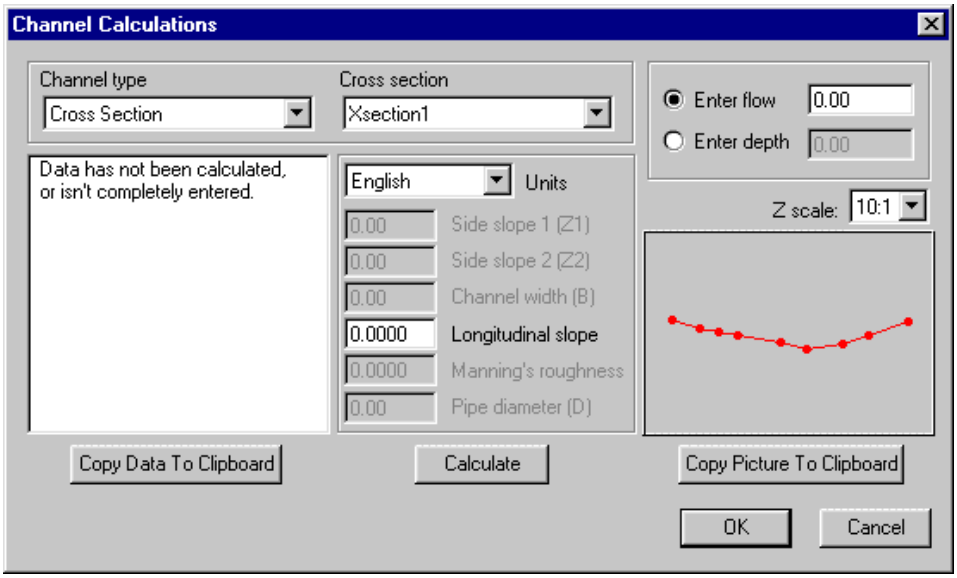


Figure 15-9 Channel Calculations Dialog

All calculations (except Froude Number) are made using Manning's Equation:

$$Q = \frac{1.49}{n} A R^{2/3} S^{1/2} \quad \text{..... 15.43}$$

where:

Q - Flow in cfs

n - Manning's roughness

A - Cross-section area of flow

R - Hydraulic radius

S - Slope

The Froude Number is computed from:

$$N_F = \frac{V}{\sqrt{gy}} \quad \text{.....15.44}$$

where:

N_F - Froude Number

V - Velocity

g - acceleration due to gravity

y - equivalent depth of flow for a rectangular channel.

The equivalent depth of flow for a rectangular channel is computed by dividing the cross sectional area of flow by the top width of the water surface.

Besides the ability to analyze the hydraulic properties of channels, results can be used to perform basic flood plain delineation. If you have computed flowrates from one of the supported hydrologic models, you can estimate stage or water surface elevation using the channel calculator. These values can then be defined at stream node locations and used as part of the flood plain delineation tools found in chapter 8. Of course most studies would require a more complete hydraulic analysis using either a 1D or 2D modeling approach such as can be done in the Surface-water Modeling System (SMS).

15.5 Weirs

Head or flow over a weir can be determined using the *Weir Calculations* dialog (Figure 15-10). If flow is to be calculated then head over the weir must be entered as an input. If head is to be computed then a flow rate must be entered as input. The weir calculator uses the standard equation for computing flow over a weir:

$$Q = C_w L h^{3/2} \quad \text{.....15.45}$$

If a hydrograph has been computed using one of the supported hydrologic model, the peak flow for the hydrograph will be used as the default flow value if the hydrograph is selected prior to opening the dialog.

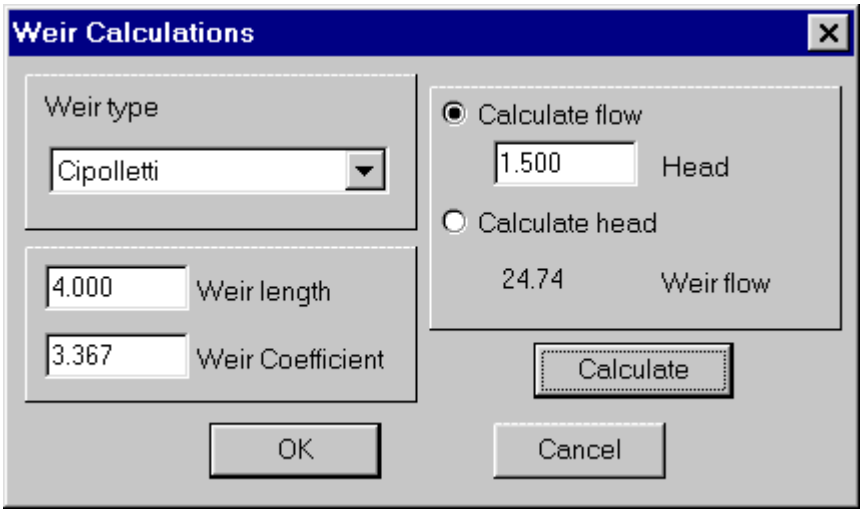


Figure 15-10 Weir Calculations Dialog

Selection of one of the predefined weir types automatically assigns the appropriate weir coefficient. A user-defined weir coefficient can also be entered, or the default value for one of the weir types listed modified. Weir calculations can also be used in combination with the detention basin calculator to define any outlet works of the basin/reservoir.

15.6 Detention Basins

An important aspect of any hydrologic study is the development of on-site storage facilities. The effects of a detention basin on an inflow hydrograph can be analyzed and an output hydrograph created in *WMS* using the *Detention Basin* calculator (Figure 15-11). This same calculator is also used in to define detention basin parameter input for *HEC-1*, the *Rational Method* and other hydrologic models as part of an overall analysis for a planned development.

A level pool routing technique is used to determine the effects of storage-routing on an input hydrograph for given detention basin/reservoir parameters. Using the principle of conservation of mass, the change in reservoir storage, S, for a given time period, Δt, is equal to the average inflow, I, minus average outflow, 0.

$$\frac{S_2 - S_1}{\Delta t} = \frac{I_1 + I_2}{2} - \frac{O_1 + O_2}{2} \quad \text{..... 15.46}$$

The defined storage vs. discharge relationships are used to iteratively solve for the end of period storage and outflow.

The detention basin calculator requires three sets of input:

1. A hydrograph.

2. A storage-capacity (volume-elevation) relationship.

3. An elevation-discharge relationship.

The way in which these three inputs are defined from the detention basin calculator is documented in the next three sections.

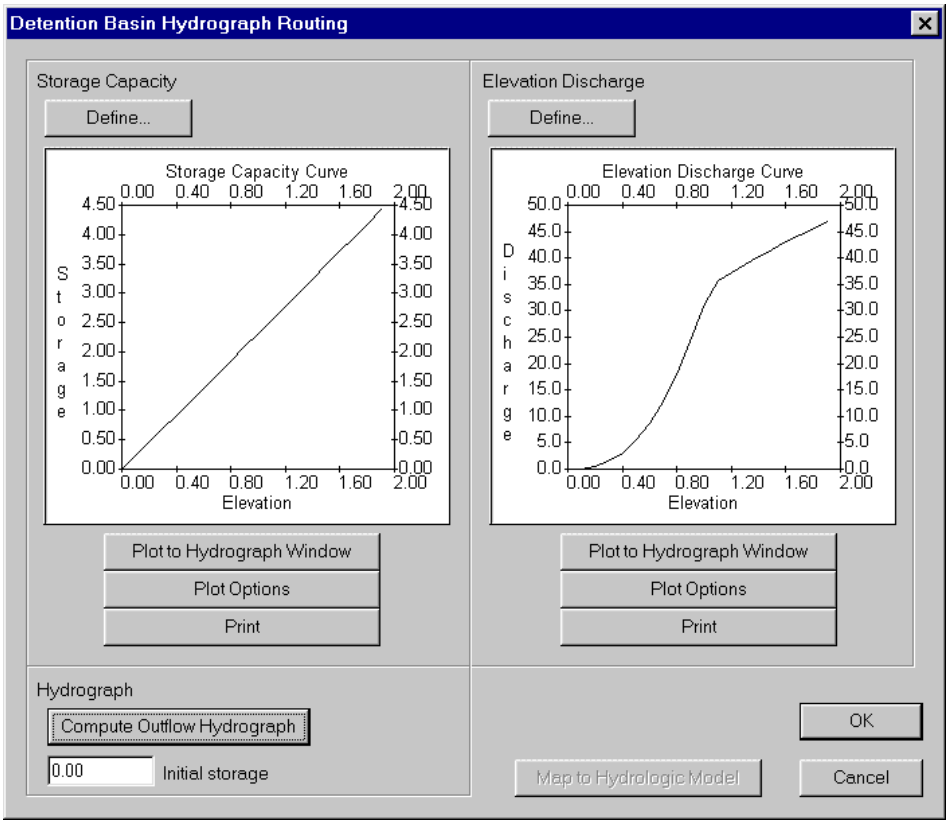


Figure 15-11 Detention Basins Dialog

When computing an outflow hydrograph an initial storage is used to account for any volume of water that may be in the detention basin prior to the arrival of the inflow hydrograph. If depth or elevation is known then the elevation vs. volume storage capacity curve must be used to determine the initial storage. The units of the initial storage should be the same as the units defined for the storage-capacity relationship

The storage-capacity and elevation-discharge curves (no matter how they are defined) are plotted in the detention basin calculator. They can also be displayed in the hydrograph window by selecting the respective *Plot to Hydrograph Window* buttons. Each of the curves can be printed by selecting the *Print* button. The Plot Options button accesses the plot options dialog in order to allow for control in the overall appearance of the defined curves.

15.6.1 Hydrograph

A hydrograph can be computed from any of the supported hydrologic models or as imported from some other source. The hydrograph should be selected prior to bringing up the detention basin calculator.

15.6.2 Storage-Capacity Curve

There are three different methods for defining storage capacity: volume vs. elevation, area vs. elevation, or known geometry. These data are defined through the Storage Capacity Input dialog shown in Figure 15-12.

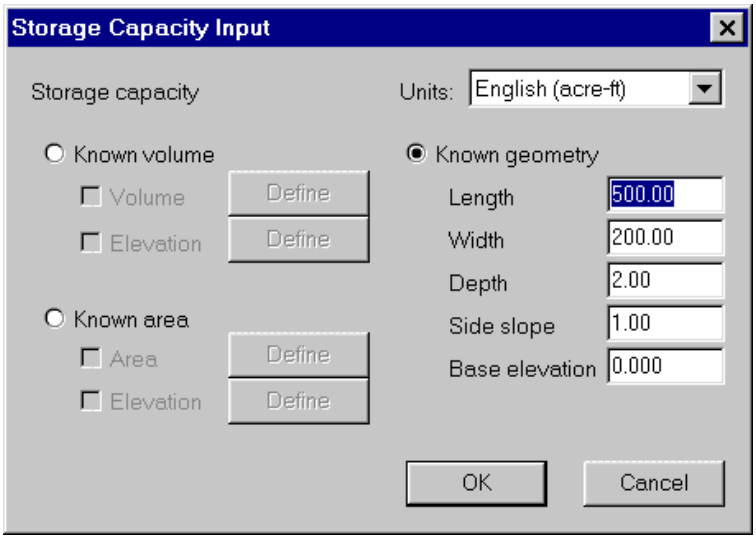


Figure 15-12

In all three cases a relationship between elevation and volume will be computed. For the volume vs. elevation option this is explicitly defined. If area vs. elevation is specified, then a corresponding volume for each elevation is computed using the conic method. The conic method is illustrated in Figure 15-13.

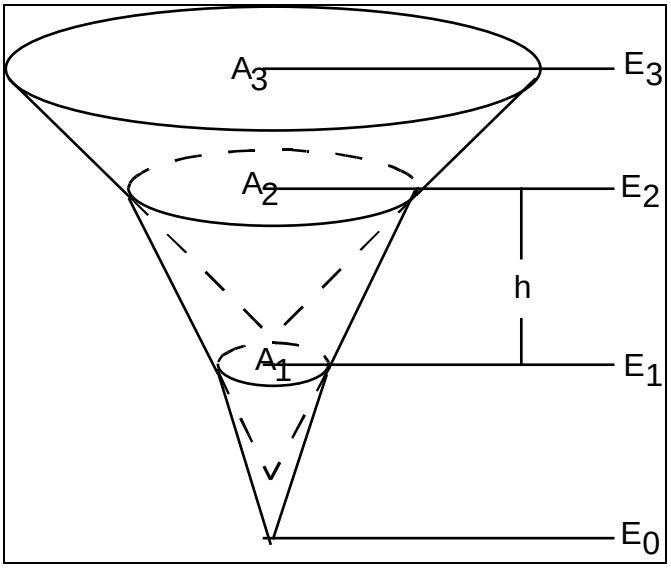


Figure 15-13 Conic Method for Volume Computations

The volume between incremental areas A_1 and A_2 is computed using the following equation:

$$\Delta V_{12} = \frac{h}{3}\left(A_1 + A_2 + \sqrt{A_1 A_2}\right) \quad \text{.....15.47}$$

where:

ΔV_{12} - The volume between areas A_1 and A_2.

A_i - surface area i.

h - vertical distance (E_2-E_1) between surface areas A_1 and A_2.

E_i - elevation of surface area i.

The same equation is used to compute the volume between each adjacent set of surface areas, with the bottom area assumed to be 0. A TIN can be used to automatically create and store for use in the detention basin calculator the elevation-volume relationship. See section 7.6.1 for details on how this can be done.

If the basin geometry option is chosen then an elevation vs. volume relationship is computed directly from the geometry defined for the basin.

15.6.3 Elevation-Discharge Relationship

Discharge data for the basin/reservoir can be entered either by supplying an elevation vs. discharge pairs, or by defining any number and combination of spillways (weirs), outlets (orifices), and standpipes (weir-orifice combinations). The *Elevation Discharge Input* dialog (Figure 15-14) is used

to set up the discharge data. If the *Known-Discharge* option is chosen then you will need to enter a series of *Elevation* and *Discharge* values (you need the same number of values in each series) to define the relationship. If the *Discharge Structures* option is chosen you can add any number of weirs, outlets, and standpipes along with their individual parameters. *WMS* will then compute an elevation discharge relationship with an appropriate elevation step and display it in the detention basin calculator. When defining a weir the parameters can be chosen from the available weirs in the weir calculator by selecting the *Weir Calculator* button.

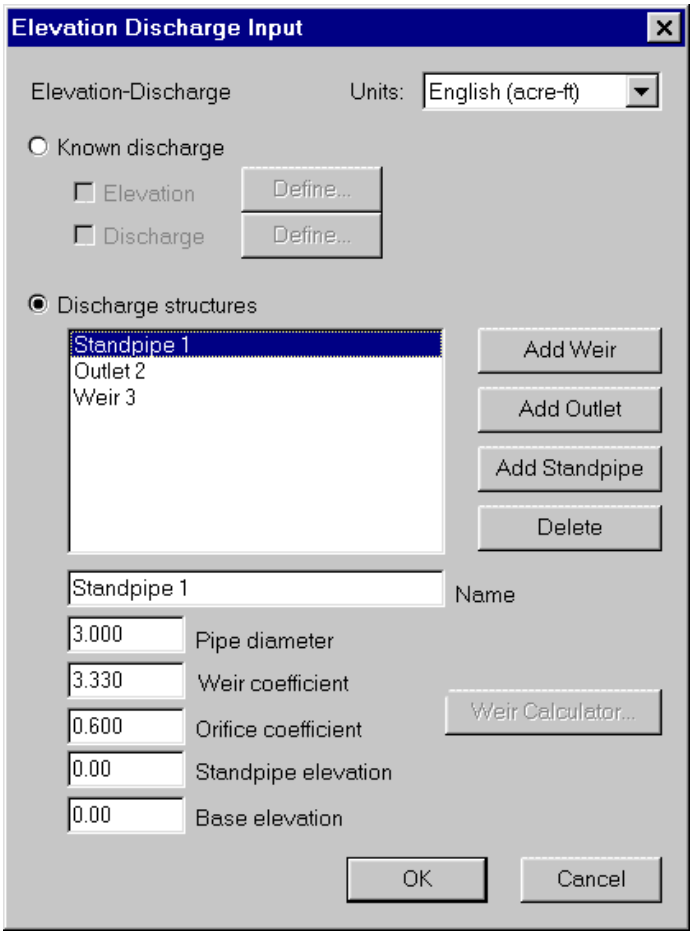


Figure 15-14

15.7 Culvert Analysis with HY8

HY8 is a BASIC program developed by the Federal Highways Administration and used to evaluate hydraulics of a culvert system. Results (peak flows) computed by one of the hydrologic models supported in *WMS* can be used to design/evaluate different sizes and combinations of culverts. *WMS* contains a graphical user interface to the HY8 program for most of the frequently used

options. In order to evaluate irregularly shaped culverts and other options not supported by the *WMS* user interface you will need to run the HY8 program outside of *WMS*. The installation of *WMS* includes a full installation of the HY8 program and is contained in the HY8 directory (this directory can be found in the same directory where WMS is installed). Because of the way the HY8 program is compiled all HY8 input/output files created by *WMS* will be stored in this directory.

This section of the *WMS* reference manual describes the user interface to HY8 while more detailed explanations can be found in the HDS-5, "Hydraulic Design of Highway Culverts," HEC-14, manual describing the methods automated by HY8.

15.7.1 HY8 General Specifications and Limitations

The dialog that controls the primary input for HY8 is shown in Figure 15-15 and can be used to specify the following culvert hydraulic calculations:

- Single culvert analysis - produces an output rating curve for a single culvert that assumes no overtopping.
- Minimize culvert width - varies the width of the first culvert and computes headwater for the design discharge until a user-defined headwater is not exceeded or either the maximum or minimum sizes are reached.
- Multiple culverts with overtopping - computes the part of flow through each of up to six culverts and over the roadway.

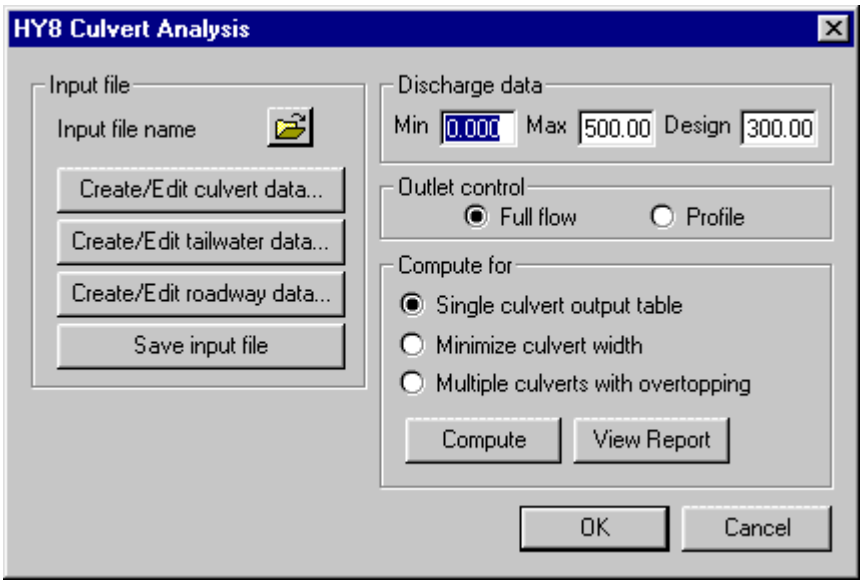


Figure 15-15 HY8 Culvert Analysis Dialog

The process of analyzing hydraulics for a culvert using the HY8 program in *WMS* consists of the following steps:

1. Enter the minimum, maximum, and design discharges.
2. Enter the information for the culvert(s) you wish to analyze (see section 15.7.2)
3. Define the tailwater rating curve (see section 15.7.3).
4. Define the roadway surface information (see section 15.7.4).
5. Save the input file (see section 15.7.5).
6. Executing HY8 and Viewing Results (see sections 15.7.6 - 15.7.8).

All of these data are accessed/defined from the *HY8 Culvert Analysis* dialog shown in Figure 15-15. If you wish to run options of HY8 not yet supported by the *WMS* interface then you can open a DOS window, change your default directory the HY8 directory found where *WMS* is installed, and type HY8 at the prompt.

15.7.2 Culvert Data

You may define up to six different culverts using the *HY8 Culvert Data* dialog shown in Figure 15-16. For each culvert you will need to specify the upstream and downstream site information. Site information may be entered as a set of stations (this will define the length of the culvert) and elevations (culvert invert data), or as a set of stations, elevations, and slopes (embankment toe data).

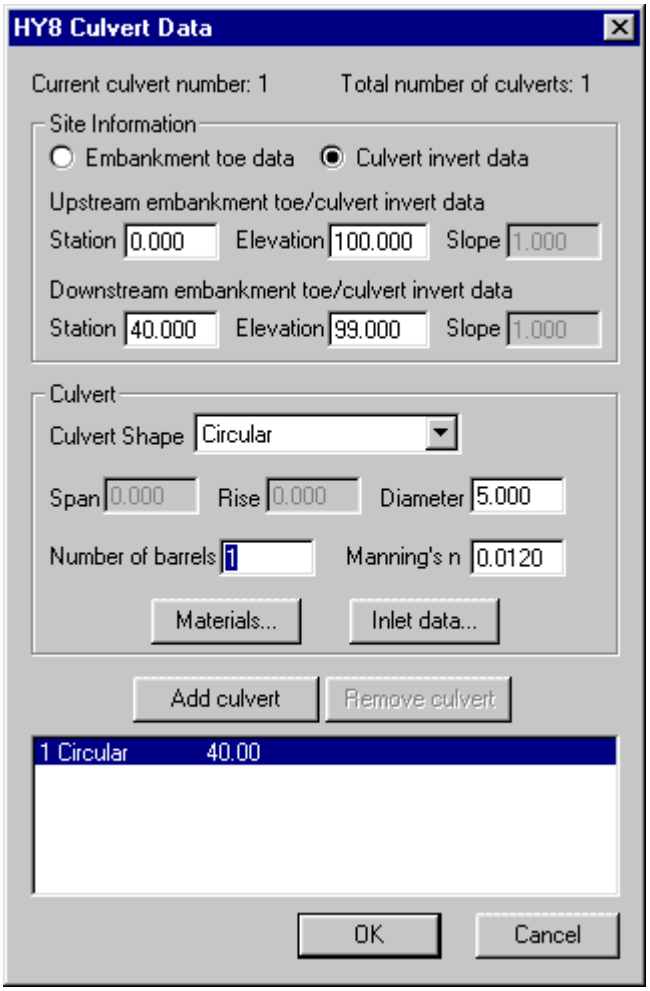


Figure 15-16 HY8 Culvert Data Dialog.

The only culvert shapes currently available in the *WMS* interface are the circular and box shapes. HY8 allows you to define elliptical and arch culverts but in order to specify these shapes you will have to run the program outside of the *WMS*.

Culvert materials are defined by selecting the *Materials…* button and selecting the desired material from the *HY8 Culvert Material* dialog shown in Figure 15-17. The available materials from which you may choose will correspond to the shape of culvert specified. By selecting a material you will actually be specifying a Manning's n or roughness coefficient. The coefficient will be updated appropriately (and can be changed if you want/need to) in the *Culvert Data* dialog after specifying the material.

Figure 15-17 HY8 Culvert Material Dialog.

Culvert inlet specifications are defined using the *HY8 Culvert Inlet* dialog shown in Figure 15-18. The available inlet types are a function of the culvert shape and material. Some of the inlet types have hydraulic improvement data that may specified, and if an inlet depression exists it can also be defined.

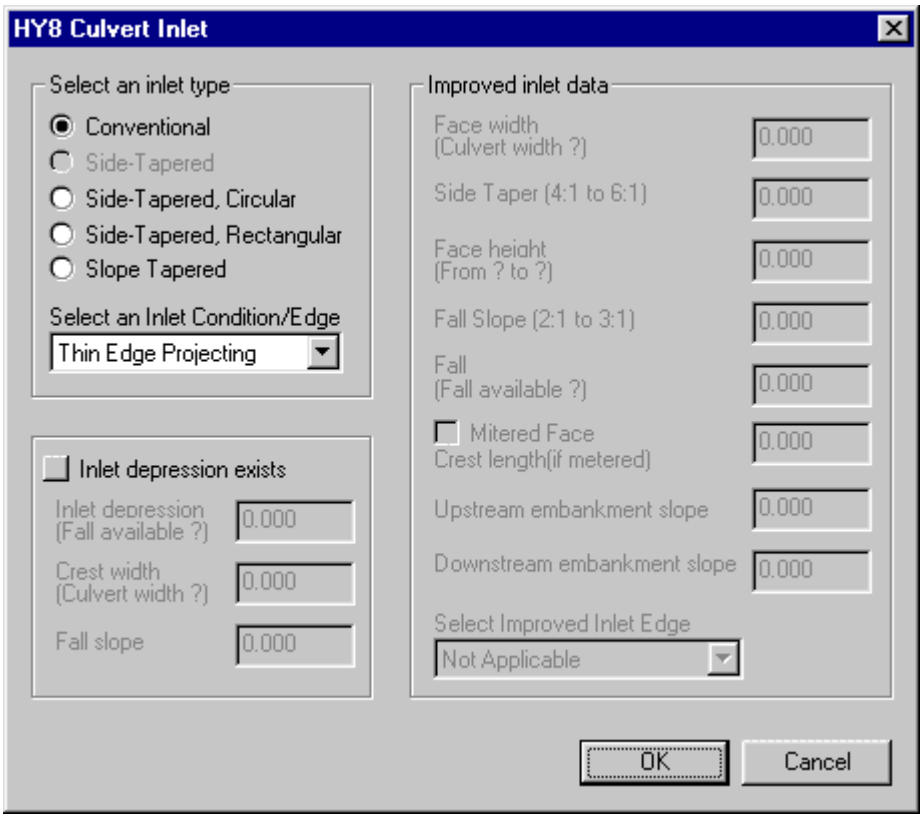


Figure 15-18 HY8 Culvert Inlet Dialog.

15.7.3 HY8 Tailwater Data

A tailwater rating curve is required as part of the culvert hydraulic computations in HY8. The rating curve can be defined explicitly if already

computed for the reach just below the culvert location, or channel geometry (exact or approximate) can be entered and a tailwater rating curve computed using the *HY8 Tailwater Data* dialog shown in Figure 15-19.

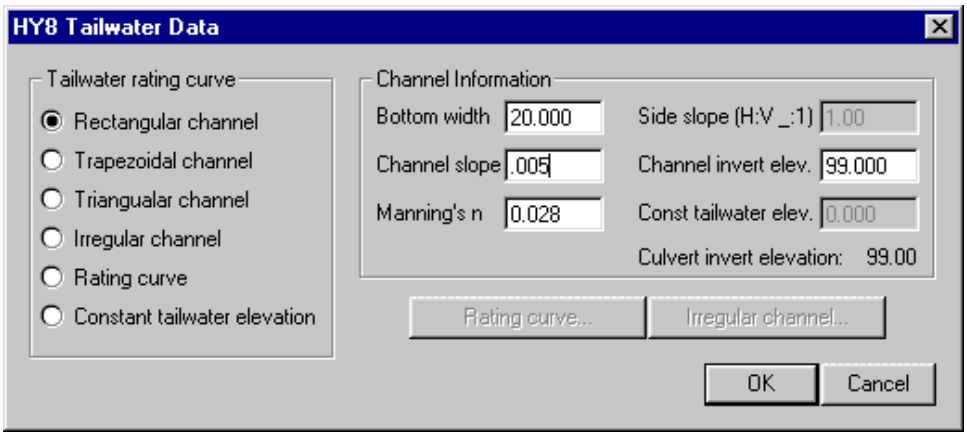


Figure 15-19 HY8 Tailwater Data Dialog.

15.7.4 HY8 Roadway Data

The roadway data defined using the *HY8 Roadway Data* dialog shown in Figure 15-20 includes information about the profile, crest and surface of the road for which the culvert is being analyzed. HY8 allows you to enter a constant elevation (the elevation should be referenced to the culvert invert elevation defined as part of the culvert data) or an irregular shape using from three to fifteen coordinates. The crest information in conjunction with the surface condition (weir coefficient) completes the information needed by HY8 to perform the "weir" calculations when water overtops the roadway surface.

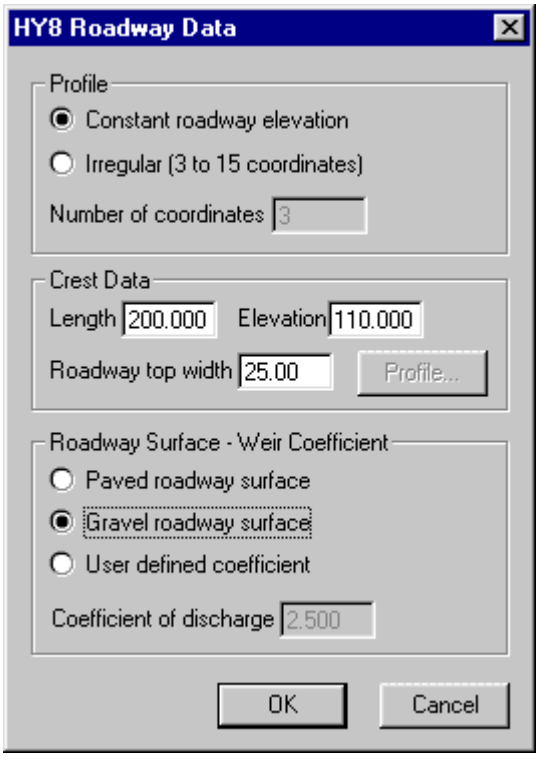


Figure 15-20 HY8 Roadway Data Dialog.

15.7.5 Saving the Input File

Once the HY8 data have been entered you must save the input file using the *Save input file* button in the main dialog (see Figure 15-15). You will not have the opportunity to specify a directory since the current implementation of HY8 in *WMS* requires that the input file be saved in the same directory where the HY8 executables reside. Once *WMS* is installed the HY8 directory will be created in the main WMS directory (the same directory where the *WMS* executable resides).

The input file name should be saved with a *.inp* extension. If you do not save the input file prior to computing the culvert hydraulics you will be prompted to do so. You may also read in an existing input file using the file browser button found near the top of the main dialog.

15.7.6 Single Culvert Analysis

When the *Compute* button is chosen with the single culvert analysis option, HY8 will compute a performance curve for the first culvert defined. When running the single culvert analysis you will see a DOS window appear and then disappear. This is normal execution and to view the output or report generated by HY8 select the *View Report* button (see Figure 15-15). The report actually contains several screens that come up in a DOS window,

including the performance rating curve as shown in Figure 15-21. The different screens in the report are scrolled by selecting the *Enter* key. The report screen also includes options for sending the report information to a text file or the printer.

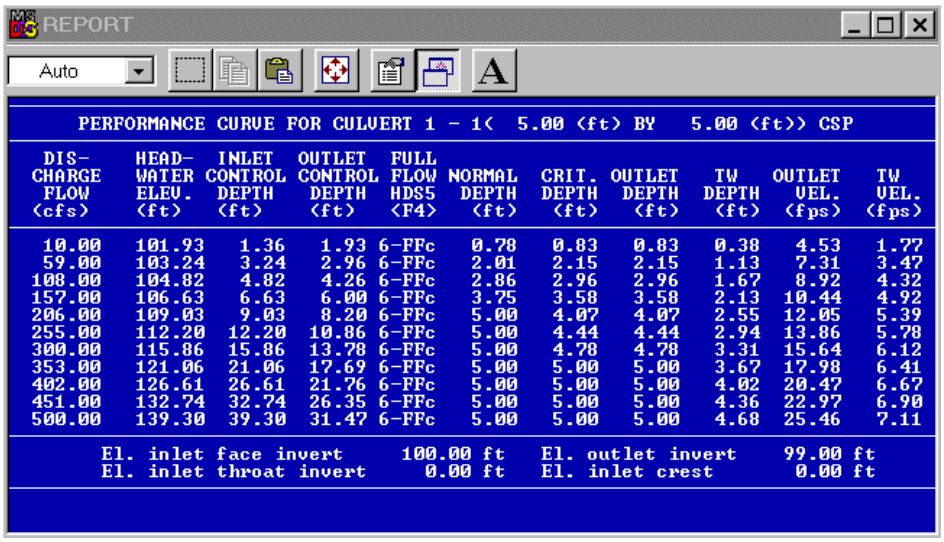


Figure 15-21 Report from a Single Culvert Analysis in HY8.

15.7.7 Minimizing a Culvert Width for a Specified Head

The second option available for computing is to minimize the culvert width for a specified head. When selecting the *Compute* button with this option specified you will see the DOS window shown at the top of Figure 15-22. You will need to enter the allowable head for which you wish HY8 to compute a culvert width that accommodates the defined discharge without overtopping. Once you enter the head value select the *Return* key and HY8 will perform the analysis and display the results as shown in the bottom of Figure 15-22.

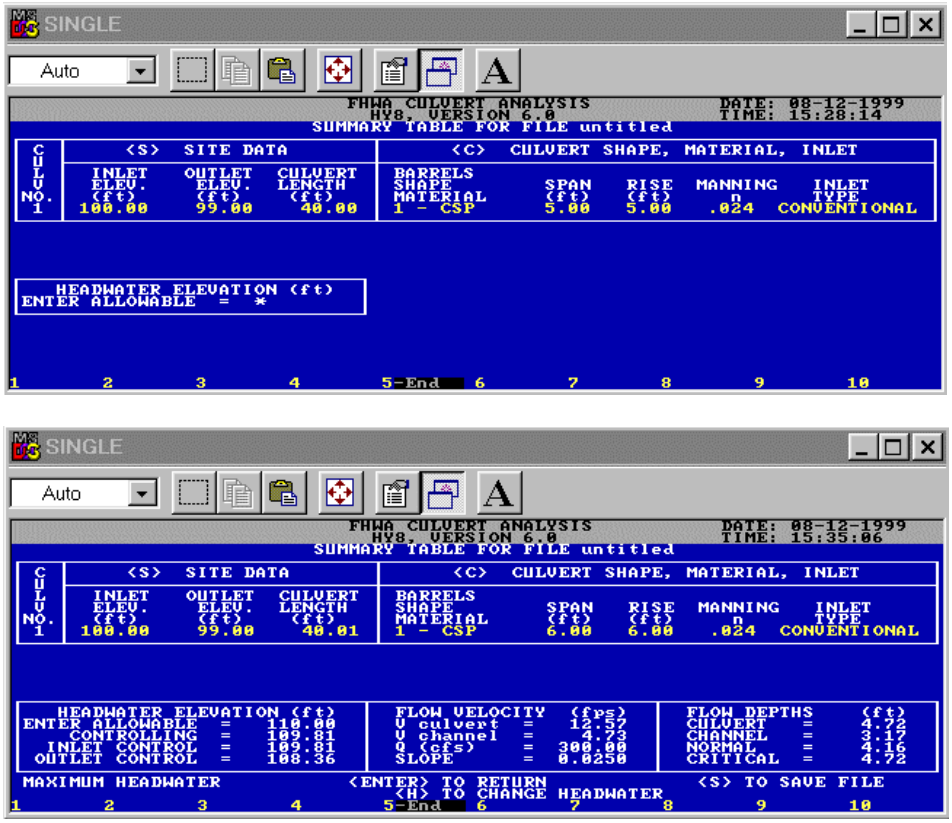


Figure 15-22 Computations for Minimizing Culvert Width.

15.7.8 Multiple Culvert Analysis and Overtopping

HY8 will also allow you to compute the hydraulics of multiple culverts. When choosing the *Compute* button with this option specified HY8 determines the amount of discharge through each of the culverts as well as the discharge overtopping the roadway. When computing with the multiple culvert option you will see the DOS window shown in Figure 15-23. In order for the balance.exe program to finish its computations you must select the Enter key. Once balance completes the computations you can view the full report (on the screen, or sent to a text file or the printer) by selecting the "*R*" key as directed by the menu at the bottom of the calculations DOS window. You can also get to the report after dismissing the balance DOS window by selecting the *View Report* button from the main dialog.

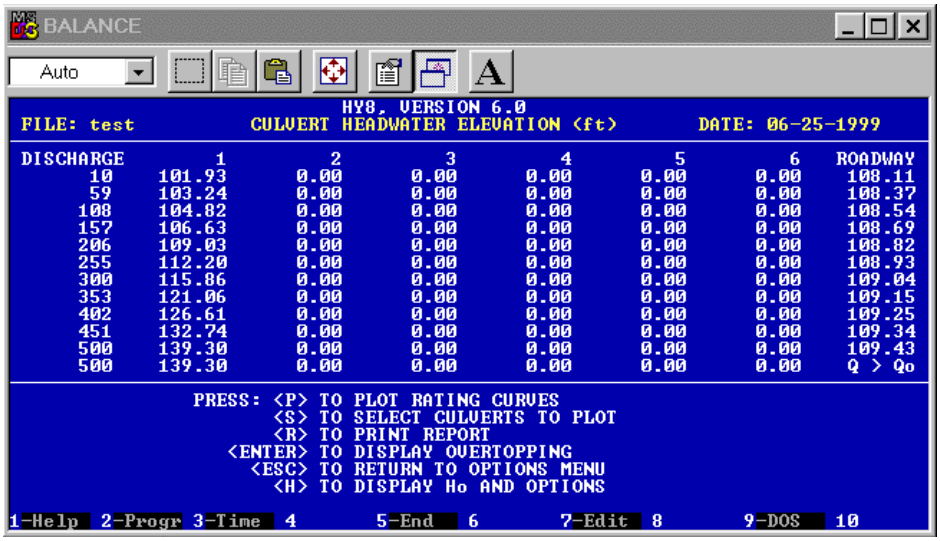


Figure 15-23 HY8 Balance Run/Report Dialog.

CHAPTER 16

Data Sets

16.1 Introduction

WMS was designed as a general purpose watershed modeling system. One of the main purposes of *WMS* is to provide a consistent interface for a variety of models and grid types. In order to accomplish this goal, input data for models and solution data (other than those models defined using topological trees) are handled in a simple, consistent fashion using data sets.

A data set is a set of values associated with each grid cell, or scatter point. A data set can be steady state (one value per item) or transient (one value per item per time step). The following objects in *WMS* each have a scalar data set list:

- 2D Grids
- 2D Scatter Point Sets

The commands for manipulating data sets are located in the *Data* menu. The data menu is one of the standard menus and is available in each of these two modules.

Data sets are used for both pre- and post-processing of models. For example, a scalar data set associated with a 2D grid can represent starting values of surface depth or values of hydraulic conductivity for a runoff modeling problem. Another data set associated with the same grid may represent computed depth values. All data sets can be used to generate contours, color

fringes, and animation sequences. A detailed discussion of how data sets can be used in *WMS* is presented in this chapter.

One advantage of the data set list approach for managing information is that it facilitates transfer of information between different types of models or models with differing resolution. This is accomplished through scatter point sets and interpolation. TINs and grids can both be converted to a 2D scatter point set. When an object is converted to a scatter point set, all data sets (or elevations in the case of TINs) associated with the object are copied to the new scatter point set. The data sets can then be transferred from the scatter point set group to other objects of any type using one of the supported interpolation schemes.

16.2 Data Browser

Most of the interaction with data sets is accomplished with the *Data Browser* (Figure 16-1). The *Data Browser* is activated by selecting the *Data Browser* command in the *Data* menu. The list box in the browser contain the lists of scalar data sets for the current object. In the case of the *2D Scatter Point* module, the data sets shown in the browser correspond to the active scatter point set. In the case of grids, there is only one grid per model and the data sets correspond to the grid associated with the current model.

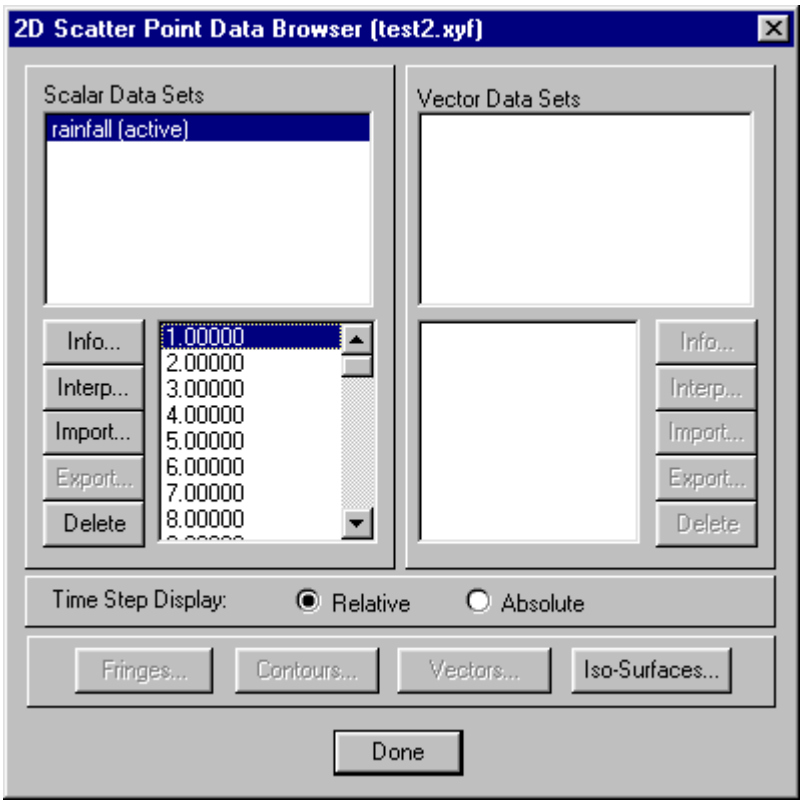


Figure 16-1 Data Browser Dialog.

16.2.1 File I/O

Previously defined data sets can be input to *WMS* by selecting the *Import* button in the *Data Browser*. This will bring up the file browser or a dialog with a list of file type options. The file types that can be imported to *WMS* as data sets are as follows:

- *WMS* ASCII Data Set Files
- *WMS* Binary Data Set Files
- *ARC/INFO*® ASCII Grid Files
- GRASS Grid Files

Additional formats will be added as new computational models are supported.

The format for the *WMS* data set files is described in Chapter 20. Once one of the file type options has been chosen, a file browser dialog appears and the user must select a file corresponding to the type selected.

Data sets can be exported from *WMS* to files by selecting the *Export* button in the *Data Browser*. Data sets can be saved as either binary or ASCII data set files. Scatter point files can also be saved from the 2D scatter point module.

When a data set is imported to *WMS*, a copy of the data set is written to a temporary file on disk in binary form. If the imported data set is already in the form of a *WMS* binary data set file, a copy of the file is not made. When part of the data set is needed it is loaded from the hard disk into internal memory. Only one time step of one scalar data set is read into internal memory at any given time. This method of file manipulation reduces the amount of RAM required, but it requires extra hard disk space. It also requires that write permission is active in the *WMS* working directory.

When a new data set is created through interpolation or using the data calculator, a temporary binary file is created for the data set. To save the data set to disk permanently, the user must select the *Export* button from the *Data Browser*.

16.2.2 Active Data Set

One data set is always highlighted in the scalar data set list. In addition, if a transient data set is highlighted, the time steps for the data set are listed in the text box directly beneath the list of data sets and one of the time steps is highlighted. The highlighted set is the active data set for the object. The values corresponding to the active data set and time step are used whenever contour, or color fringe plots are generated. In addition, the entire range of

time steps of the active data set are used whenever animation film loops are generated.

Whenever a new data set is created by importing from a file, interpolating, or using the data calculator, the data set becomes the active data set for the object.

16.2.3 Elevations

Whenever a grid or scatter point set is created or read from a file, a scalar data set is created containing the elevations of the cells or data points. Thus, there is always at least one data set associated with each grid. This data set cannot be deleted.

16.2.4 Deleting Data Sets

Data sets can be deleted by selecting the data set in the list box and selecting the Delete button in the *Data Browser*. This deletes the binary copy of the data set on disk. If the original data set file was already in binary form, the file is not deleted.

All data sets associated with an object are automatically deleted whenever the object is deleted or whenever the number of cells or vertices in the grid or scatter data set is changed due to an editing command.

16.2.5 Data Set Info

The *Info* buttons in the *Data Browser* will bring up a dialog listing some of the main characteristics of the active scalar data set. These include statistics such as maximum, minimum and range as well as mean and standard deviation. The name of the active data set can also be edited from the *Info* dialog.

16.3 Data Calculator

The *Data Calculator* can be used to perform mathematical operations with data sets (Figure 16-2). The *Data Calculator* can be accessed by selecting the *Data Calculator* command from the *Data* menu.

The *Data Calculator* can be used to perform any of the mathematical operations shown as icons in the center of the dialog. Some of the operators are binary (ex., "+", "-") and some are unary (ex., "1/x", ln(x)). When a binary operator is selected, either two data sets or a data set and a scalar must be chosen as operands. When a unary operator is selected, one data set is chosen as the operand. In either case, a name for the new data set created as a result of the operation must be entered. Once the mathematical operation is

completely defined, the user clicks on the "=" button to execute the operation. If either of the "Delete" toggles are set, the data set corresponding to that "Delete" toggle is deleted after the operation is completed.

The *Data Calculator* is useful for a variety of tasks. For example, to generate a data set representing the absolute difference between two other data sets, the user first subtracts one data set from the other to make a new temporary data set. The absolute value unary operator is then applied to this temporary data set to get the final data set representing the difference between the two original data sets. Such a data set would be useful for comparing the results of two separate solutions computed by a numerical model.

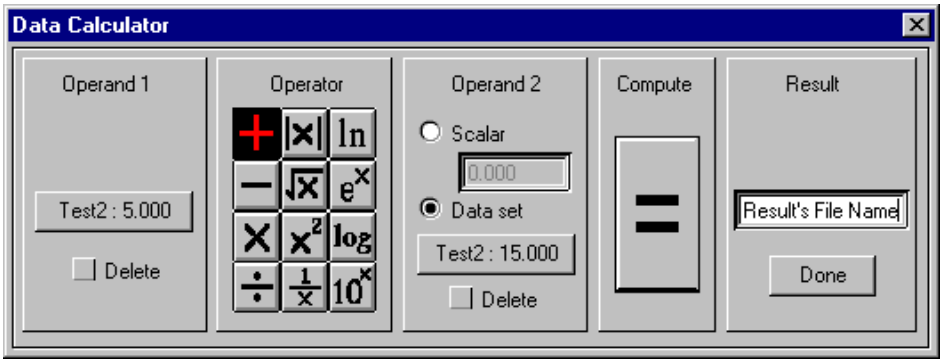


Figure 16-2 Data Calculator Dialog.

16.4 Contours

Most of the objects supported by *WMS* can be contoured by turning on the contour option in the *2D Grid* tab of the *Display Options* dialog. When a grid is contoured, the scalar values associated with the active data set for the grid are used to generate the contours. Since contouring can be done for several of the different modules, the discussion of contouring options is given in section 2.10.2.

16.5 Fringes

If the *Fringes* item in the *Display Options* dialog is selected for an object, a color-shaded image will be generated when the object is shaded. The current color ramp and the values of the active scalar data set are used to vary the colors on the grid in a continuous fashion.

By default, the minimum color on the color ramp is associated with the minimum data set value and the maximum color is associated with the maximum data set value. The *Fringe Options* dialog can be used to force the ramp of colors to be confined to a smaller interval specified by the user. This forces all of the color gradation to be concentrated in a particular range of

interest. The *Fringe Options* dialog is accessed using the *Fringe Opts* command in the *Data* menu or from the *Data Browser*.

16.6 Mapping Elevations

For 2D grids it is often useful to change the values used for the elevations of the objects. For example, suppose a set of data values has been interpolated to a grid. The values can be displayed using contours. Another way to display the values is to map the data set to the mesh elevations. This option further emphasizes the variation in the data when the grid is displayed in oblique view.

Any data set can be mapped to elevations using the *Map to Elevations* command in the *Data* menu. The original elevations are always saved as a data set so that the original elevations can be restored at a later time.

16.7 Film Loop Animation

One of the most powerful 2D visualization tools in *WMS* is animation. An animation sequence can be generated for a grid with a transient data set to illustrate how contours, vary as a function of time. Each frame of the animation is stored as an image as part of an AVI file. The entire set of frames in an animation sequence is referred to as a film loop.

Animation film loops are generated by selecting the *Film Loop* command in the *Data* menu. This command brings up the *Film Loop* dialog shown Figure 16-3. The *Film Loop* dialog is used to control the playback of film loops. A new film loop can be generated by selecting the *Setup* button. Once a film loop has been generated, it can be saved to an AVI file using the *Save* button. Previously saved film loops can be read from disk using the *Read* button (they can also be run using any AVI playing software, or included in presentation software documents).

Figure 16-3 Film Loop Dialog.

16.7.1 Saving Film Loops

Saving and reading film loops is useful since some film loops may take a significant amount of time to generate depending on the complexity of the image. The film loops are saved to disk in a compressed AVI format.

16.7.2 Film Loop Playback

Once a new AVI film loop has been generated or an AVI film loop has been read from disk, several options are available for playing back the film loop. The buttons at the upper left of the *Film Loop* dialog are designed to mimic the buttons on a VCR or CD player. The Play button causes the film loop to cycle continuously. The *Stop* button halts the playback. The *Step* buttons can be used to advance the film loop forward or backward one frame at a time. In addition, the frame scroll bar can be used to interactively move the frames forward or backward.

The speed of playback can be adjusted using the *Speed* scroll bar. The maximum speed depends on the speed of the computer and the size of the image being animated. The smaller the image, the faster the maximum playback speed.

Two options are available for cycling the film loop playback. The *continuous playback* option starts a new cycle at the first frame in the loop after the last frame is encountered. The *oscillation* option plays the loop in the forward direction to the end of the loop and then in the reverse direction back to the beginning of the loop.

16.7.3 Film Loop Setup

A new film loop can be generated by selecting the *Setup* button in the *Film Loop* dialog. This button accesses the *Film Loop Options* dialog (Figure 16-4).

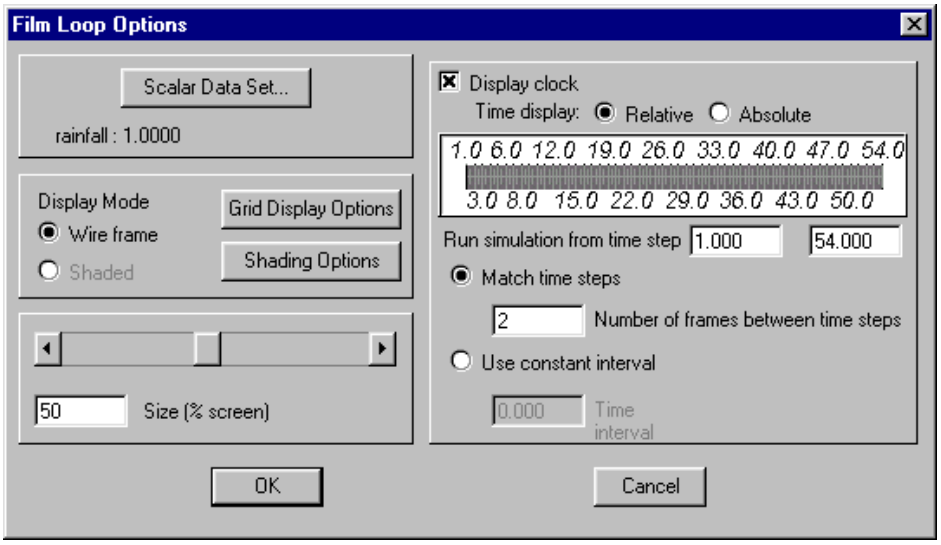


Figure 16-4 Film Loop Options Dialog.

Data Set

Film loops are always generated using the active data set. The *Scalar Data Set* button at the top of the dialog can be used to change the active scalar data set. The current active data set is displayed to the right of this button.

Display Mode

The display mode is used to control whether each frame is generated as a wire frame image or a shaded image using the current shading options.

Image Size

By default, each frame that is generated in a film loop occupies the entire *Graphics Window*. This results in film loops composed of large images which require a significant amount of memory and which are difficult to playback at a high speed. To reduce the size of the film loop, the individual frames can be generated at a specified fraction of the default size. The memory required for a film loop is quadratically proportional to the fractional size. For example, an image generated at 50% of the *Graphics Window* size requires 25% as much memory as an image generated at full size.

Transient Animation

Transient animation can be used with 2D grids and a transient data set. As each frame is generated, a set of values corresponding to the current time is loaded into memory and the image is redrawn using the current display options. Thus, if the contour display option is selected, the contours will vary from frame to frame.

The strip on the right of the transient animation section of the *Film Loop Setup* dialog is used to specify what range of the available time steps are to be used for animation. The range of time steps can also be entered directly in the edit fields below the time step strip. The range displayed in the strip corresponds to the scalar data set.

The total number of frames generated in the film loop can be defined by either matching the time steps (one frame per time step) or by using a constant interval (e.g., one frame for every two hour interval). If the *Match Time Steps* option is chosen, extra frames can be created between each time step if necessary using linear interpolation of the data values at the specified time steps.

16.8 Gages

One of the most important steps in any modeling problem is calibration. During the calibration phase, an attempt is made to model a set of conditions which have been known to exist at a site and for which measured data (surface depth, infiltration) are available. The geometry, resolution, and input parameters of the model are adjusted until the output computed by the model is reasonably close to the measured data.

The calibration stage can be the most tedious and time-consuming portion of the modeling process. In order to make the calibration stage more efficient, a set of tools for managing gages has been provided in *WMS*. A "gage" is an xyz point defined by the user representing a location where field data has been collected (ex., a gaging station) or simply a point of interest in the model. Once a set of gages has been defined, whenever a transient data set is imported to *WMS* to a grid, the data set is interpolated to each gage and a curve is drawn in the *Hydrograph* window representing the variation of the data set with time at each gage. The plot can be customized to include any combination of gages and data sets. Field data can be imported from text files and plotted for comparison with computed curves.

Gages and gage plots are supported only in the *2D Grid* module.

16.8.1 The Gages Dialog

A set of gages can be created by selecting the *Gages* command from the *Data* menu. This command activates the *Gages* dialog shown in Figure 16-5. All existing gages are listed in the text box in the upper left corner of the dialog. One of these gages is highlighted at all times. The name, color, and location of the highlighted gage can be edited using the controls on the right hand portion of the dialog. The color and name are used in displaying the gage in the *Graphics Window*.

A new gage can be added to the list of gages by selecting the *New* button beneath the list of gages. A highlighted gage can be removed from the list by selecting the delete button. A set of gages can be imported from a text file by selecting the *Import* button. The format of the gage file is described in Chapter 20. A set of measured curves can be included in the gage file for comparison with computed curves. A set of gages created within *WMS* can be exported to a file for future use by selecting the *Export* button.

The *Interpolation Method* options in the lower right corner of the *Gages* dialog controls how data sets are interpolated to the gages for curve plotting. If the *Interp. from neighboring nodes/cells* option is chosen, the data sets are interpolated from the nodes or cells in the vicinity of the gage using a simple inverse distance weighted interpolation scheme.

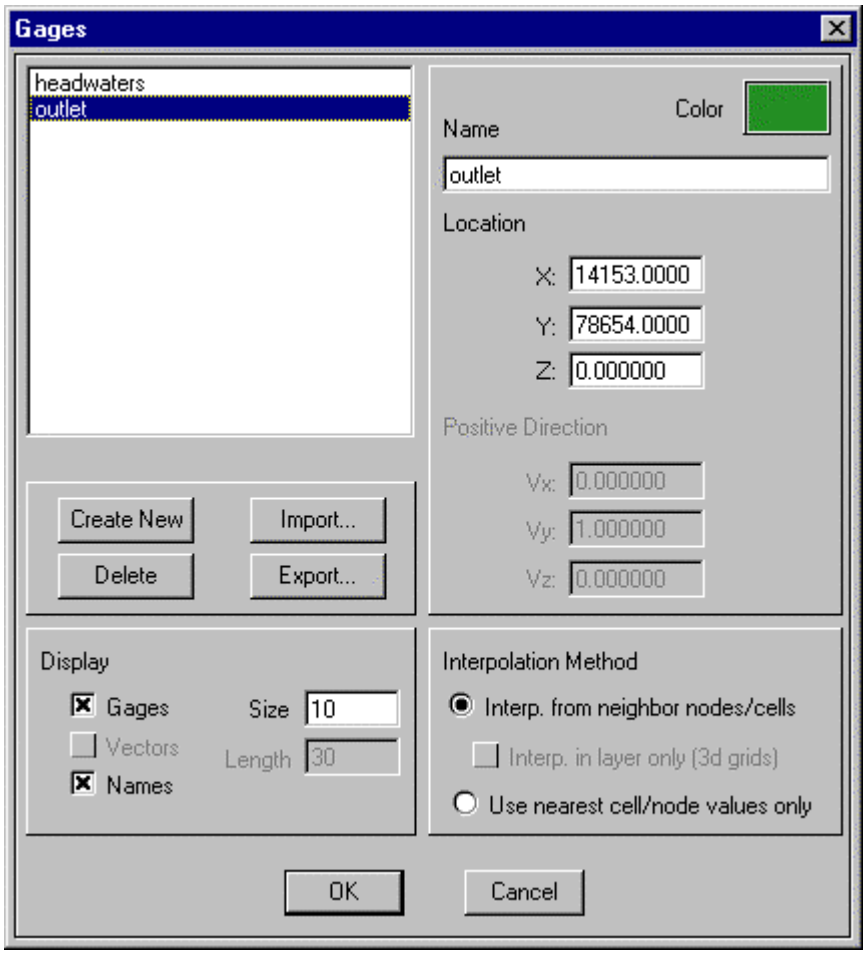


Figure 16-5 Gages Dialog.

Gages are plotted in the *Graphics Window* as shown in Figure 16-6. The name is plotted just below the gage symbol. Each component of the gage can be turned on or off or resized using the *Display* items in the lower left corner of the *Gages* dialog.

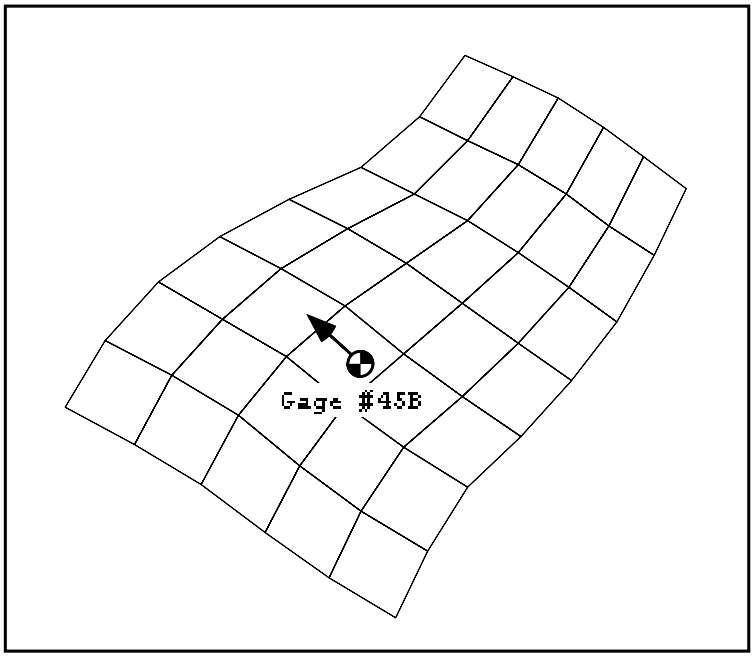


Figure 16-6 A Gage Symbol Plotted in the Main Graphics Window.

16.8.2 The Gage Tools

Gages can also be created and deleted using the *Gage* tools. The gage tools appear in the dynamic portion of the *Tool Palette* of each of the modules which support gages. The tools are as follows:

The Create Gages Tool

The *Create Gages* tool is used to interactively create gages in the *Graphics Window*. When this tool is active, a new gage is created by clicking in the *Graphics Window* at the desired location of the gage (the *Graphics Window* must be in plan view when creating gages). The xy coordinates of the gage are defined by the cursor position and the user is prompted for the z coordinate. The x, y, and z coordinates of a new gage can be edited using the *Edit* Window. In addition, once a gage has been defined with the *Create Gages* tool, the gage can be edited using the *Gages* dialog.

The Select Gages Tool

The *Select Gages* tool is used to select previously defined gages. A set of selected gages can be deleted by hitting the *DELETE* key or by selecting the *Delete* command from the *Edit* menu. The coordinates of a selected gage can be edited using the *Edit Window*. The location of a gage can also be edited by holding down the mouse button when a gage is selected and dragging the gage.

This tool is also used to control what is plotted in the *Gage Plot Window*. Only the curves associated with selected gages are plotted.

16.8.3 The Gage Plot Manager

Once a set of gages has been defined, one or more plots can be generated in the *Hydrograph Window* representing the variation vs. time of any of the transient data sets associated with grids interpolated to the gages. Up to five plots may be generated at once. Any combination of data sets can be displayed on a single plot. The curves are plotted only for gages which have been selected using the *Select Gages* tool. This makes it possible to quickly change the combination of curves plotted.

The *Gage Plot Manager* command in the *Data* menu activates the *Gage Plot Manager* dialog shown in Figure 16-7. The names of five plots are listed in the text box whether they are displayed or not. The word *visible* or *hidden* will appear beside each plot title indicating whether the plot is currently displayed in the *Hydrograph Window*. The titles of the plots are listed in the order that they are displayed. One of these plots is highlighted at all times.

To display a plot in the *Hydrograph Window*, first select the plot to be displayed in the text box, then select the *Make Visible* button. To hide the plot, select the *Hide Plot* button.

The *Gages* button in the *Gage Plot Manager* brings up the *Gages* dialog described above. This dialog can also be activated using the *Gages* command in the *Data* menu.

The *Print* button in the *Gage Plot Manager* prints a copy of the plots currently displayed in the *Hydrograph* window.

The *Export WKS* button exports a copy of the points used to create the plots in the plot window to a spreadsheet file. This allows more customization of the plots if necessary.

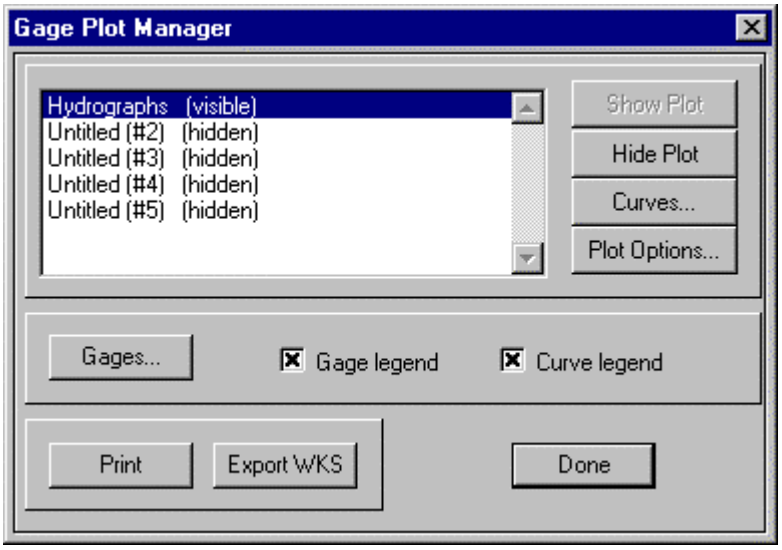


Figure 16-7 Gage Plot Manager Dialog.

The Curves Dialog

The set of curves displayed in each plot can be edited by selecting the *Curves* button in the *Gage Plot Manager*. This button activates the *Curves* dialog shown in Figure 16-8.

Two lists of curves are displayed in the dialog. One of the curves is always highlighted in each list. The list on the left represents the curves which are available for plotting. The list on the right represents the curves which are being displayed in the selected plot. Initially, the list on the right contains one curve for each data set. These curves represent the values of the data set interpolated to the gage locations. The list also contains one curve for each of the measured curves imported with a gage file. Curves in the available lists are moved to the list of plotted curves using the *Selected* -> and *All* -> buttons. The *Selected* -> button moves the highlighted curve and the *All* -> button moves all of the curves. Likewise, curves can be moved from the plotted curves list to the available curves list using the <- *Selected* and <- *All* buttons.

The display options for the highlighted curve in the plotted curves list can be edited using the group of controls at the bottom of the *Curves* dialog. The curves can be plotted by displaying a symbol at each point on the curve or by displaying a line through the points or with a combination of points and lines. The symbol, line thickness, and line style can all be edited.

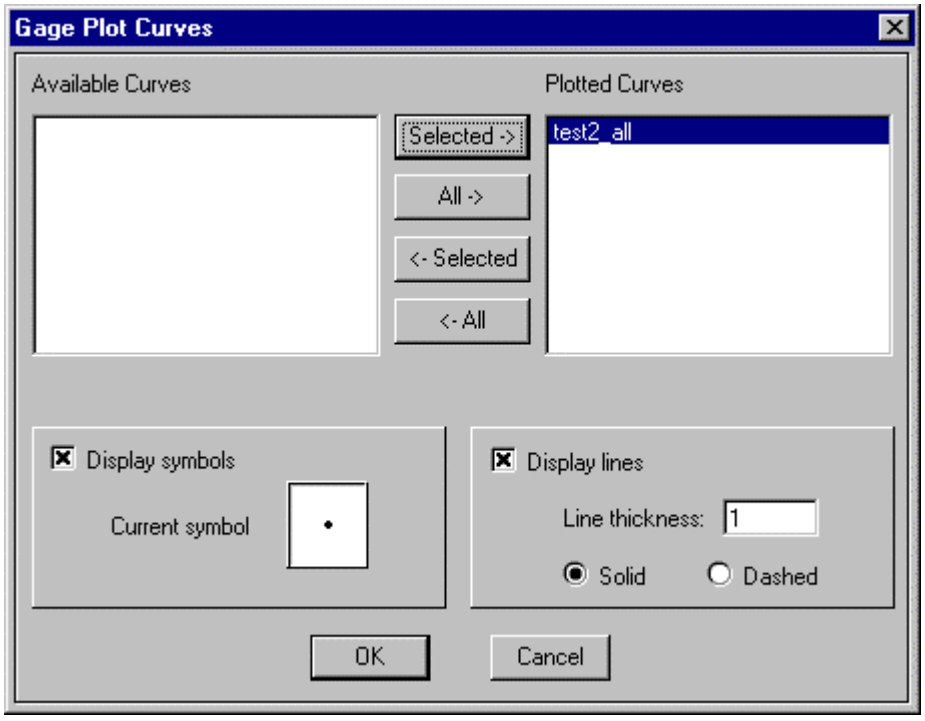


Figure 16-8 Gage Plot Curves Dialog.

The Gage Plot Options Dialog

The options used to display the plots listed in the *Gage Plot Manager* can be edited using the *Gage Plot Options* dialog. The options for an individual plot are edited by selecting the plot in the plot list and then selecting the *Plot Options* button from the *Gage Plot Manager* dialog. This activates the dialog shown in Figure 16-9.

A major and a minor title can be entered. Both titles are displayed at the top of the plot with title 2 displayed beneath title 1. The x and y titles are displayed by the x and y axes.

The *Foreground color* is used to display the titles, axes, grid lines, etc. The *Background color* is used to fill in the background of the plot. The *Curve legend* option causes a legend to be printed on the plot showing the symbol and line style used for each curve next to the curve name.

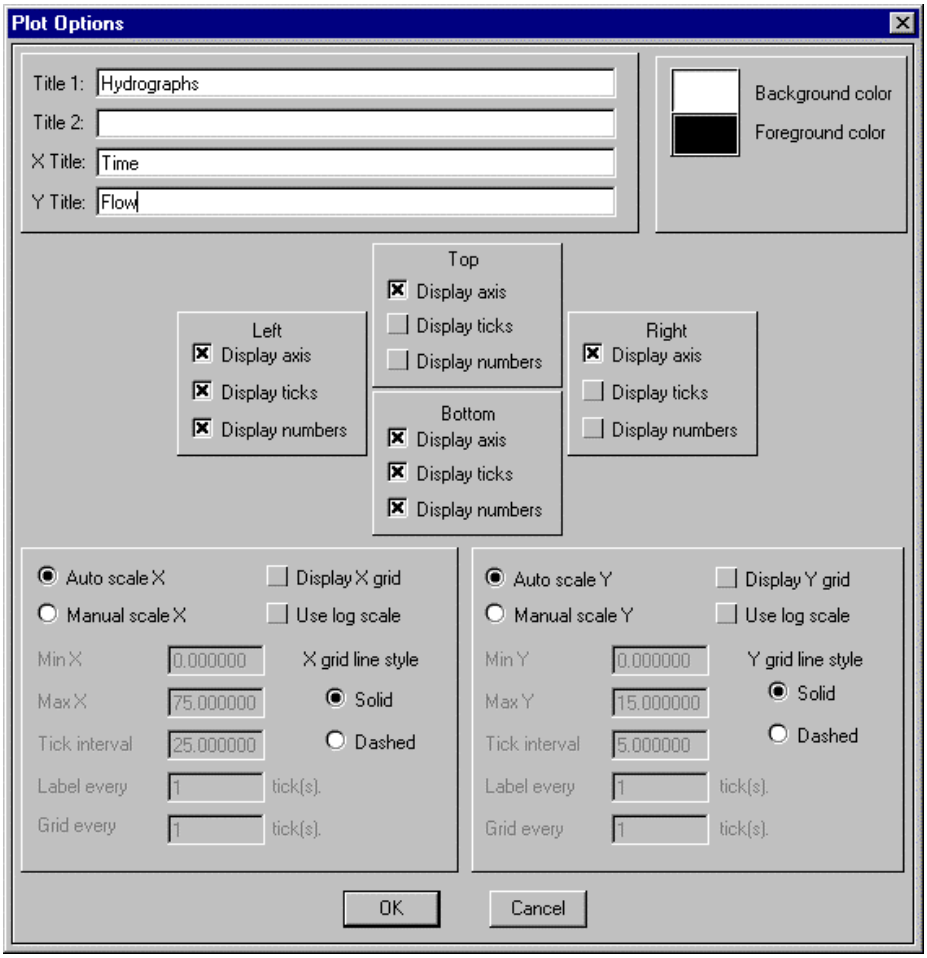


Figure 16-9 Plot Options Dialog.

The x and y axes can be displayed using either the *Autoscale* option or the *Manual scale* option. With the *Autoscale* option, the range, tick interval, label interval, and grid line interval are all chosen automatically. With the *Manual scale* option, each of the options can be specified explicitly.

CHAPTER 17

2D Grids

17.1 Introduction

The *2D Grid* module is used for surface visualization and as the primary building block of the two-dimensional finite difference surface runoff model *CASC2D*. Scalar values can be interpolated to a grid from a set of 2D scattered data points. These values can then be contoured or displayed with hidden surface removal and color fringes to display the variation in the interpolated data. The scalar values for grid cells may also be used as input parameters for running a *CASC2D*.

17.2 Grid Types

Two types of grids are supported in the *2D Grid* module: mesh centered grids and cell centered grids (Figure 17-1). When computations are performed on a mesh-centered grid, the computation points are the grid nodes or the corners of the grid cells. With a cell-centered grid, computations are performed at the cell centers.

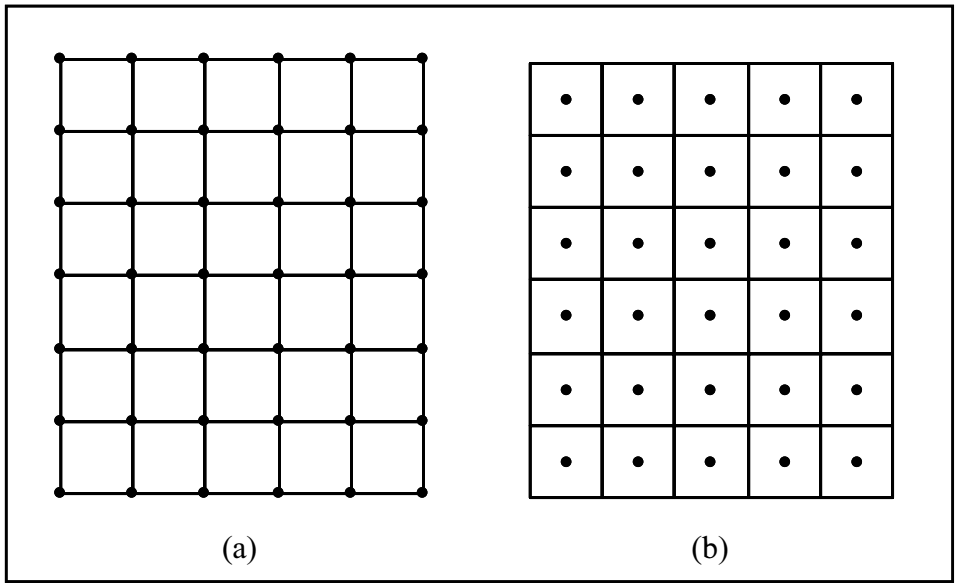


Figure 17-1 Types of 2D Grids Supported in WMS*. (a) Mesh-Centered Grid (b) Cell-Centered Grid.*

When a data set is imported to a cell-centered grid, there is one value in the data set for each cell. The contouring and fringing functions use scalar values at the cell corners. Therefore, whenever contouring or fringing is performed, the values at the cell centers are interpolated to the cell corners. Interpolation to cell corners is only done for visualization purposes. All computations performed using the data calculator are performed on the original values at the cell centers. With mesh-centered grids, all visualization and computations are performed at the cell corners and no interpolation is necessary. The *CASC2D* model requires the grid to be cell centered. However, if the purpose of the grid is for visualization of interpolated data only, then it is preferable to use a mesh centered grid since the extra interpolation to the cell corners for contouring is not required.

17.3 Tool Palette

The following tools are contained in the dynamic portion of the *Tool Palette* when the *2D Grid* module is active.

17.3.1 Select Cell

The *Select Cell* tool is used to select individual grid cells or grid nodes. Multi-selection can be performed by holding down the *SHIFT* key while selecting or by dragging a rectangle to enclose the cells to be selected. The ij indices of the selected cell are displayed in the *Edit Window*.

If the grid is cell-centered, selected cells are highlighted by drawing small circles around the cell centers. If the grid is mesh-centered, the grid nodes (corners) are selected, and the circles are drawn around the grid nodes.

Only visible cells can be selected. Cells which have been hidden cannot be selected. Inactive cells can only be selected when they are being displayed by turning on the *Inactive Cells* item in the *Display Options* dialog (see section below on active/inactive cells).

17.3.2 Select i

The *Select i* tool is used to select an entire "row" (set of cells with the same i index) of cells at once. Multi-selection can be performed by holding down the *SHIFT* key. The i index of the selected row is displayed in the *Edit Window*.

17.3.3 Select j

The *Select j* tool is used to select an entire "column" (set of cells with the same j index) of cells at once. Multi-selection can be performed by holding down the *SHIFT* key. The j index of the selected column is displayed in the *Edit Window*.

17.3.4 Contour Labels

The *Contour Label* tool manually places numerical contour elevation labels at points clicked on with the mouse. These labels remain on the screen until the contour options are changed, until they are deleted using the *Contour Labels* dialog, or until the grid is edited in any way. Contour labels can be deleted with this tool by holding down the *SHIFT* key while clicking on the labels. This tool can only be used in plan view.

17.3.5 The Create Gages Tool

The *Create Gages* tool is used to interactively create gages in the *Graphics Window*. When this tool is active, a new gage is created by clicking in the *Graphics Window* at the desired location of the gage (the *Graphics Window* must be in plan view when creating gages). The xy coordinates of the gage are defined by the cursor position and you are prompted for the z coordinate. The x, y, and z coordinates of a new gage can be edited using the *Edit Window*. In addition, once a gage has been defined with the *Create Gages* tool, the gage can be edited using the *Gages* dialog. Display of gages are described in more detail in section 16.8.

17.3.6 The Select Gages Tool

The *Select Gages* tool is used to select previously defined gages. A set of selected gages can be deleted by hitting the *DELETE* key or by selecting the *Delete* command from the *Edit* menu. The coordinates of a selected gage can be edited using the *Edit Window*. The location of a gage can also be edited by holding down the mouse button when a gage is selected and dragging the gage. This tool is also used to control what is plotted in the *Gage Plot Window*. Only the curves associated with selected gages are plotted.

17.4 Display Options

The display options control which components of the grid are displayed. The display options can be set by selecting the *Display Options* command in the *Grid* menu. Most of the items in the dialog box are toggle boxes. If the toggle for a component of the grid is set, the component is displayed when the grid is re-drawn. The color used to display the component can be set using the pop-up color window to the left of the toggle box.

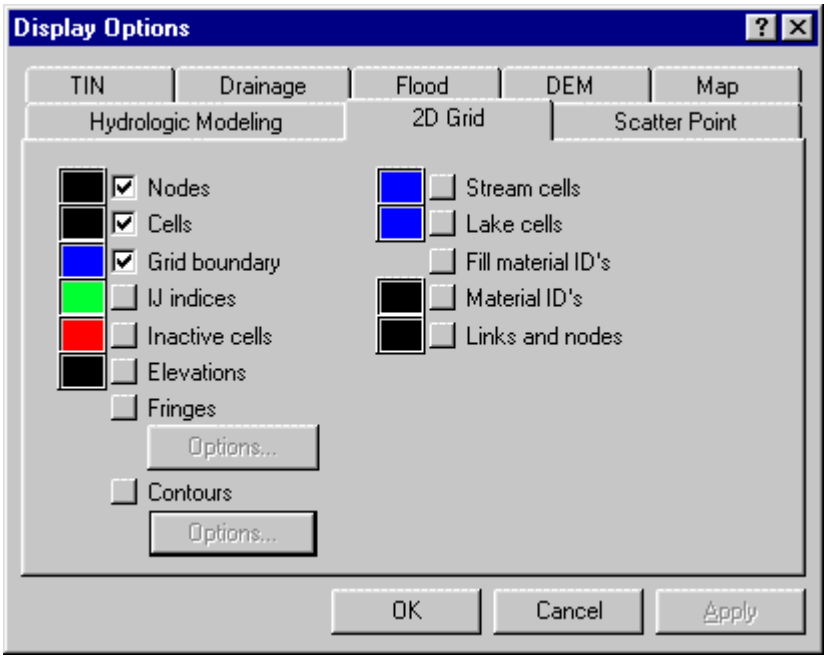


Figure 17-2 2D Grid Display Options Dialog.

The grid display options are as follows:

- The *Nodes* item is used to display grid nodes. If the grid is cell-centered, a dot is displayed at the cell centers. If the grid is mesh-centered, a dot is displayed on the cell corners.

- The *Cells* item is used to display the edges of active grid cells. The cells are drawn using the specified grid cell color.

- The *Grid Boundary* item is used to display a solid line around the perimeter of the grid. Displaying the boundary is useful when contours are being displayed with the cell edges turned off.

- The *IJ Indices* item is used to display the ij indices of each cell or node.

- If the *Inactive Cells* item is used to display cells which are inactive. If this option is turned off, inactive cells are not displayed. Inactive cells must be displayed before they can be selected.

- The *Elevations* item is used to display the z coordinate of each node or cell.

- The *Fringes* item is used to display color fringes on the grid when the grid is shaded. The active scalar data set is used to display the fringes.

- The *Contours* item is used to display contours computed using the active scalar data set.

17.5 Grid Generation

Three different methods are available for creating grids:

Create Grid	Specify the dimensions and starting location of the grid
Import a GIS Grid	Read a grid from either an *ARC/INFO*® ASCII, or *GRASS* grid using the *Import* command from the *File* menu. Importing of grid files is explained in section 2.8.6.
Feature Objects	Define a bounding polygon in the Map module and use it in conjunction with a background elevation map to generate a grid covering the extents of the polygon, with only the cells inside the polygon being active. Creating grids with feature objects is explained in section 3.2.8.

17.5.1 Create Grid

A new grid can be created by selecting the *Create Grid* command from the *Grid* menu. This command brings up a dialog (Figure 17-3) which allows the user to specify the x and y bounds of the grid.

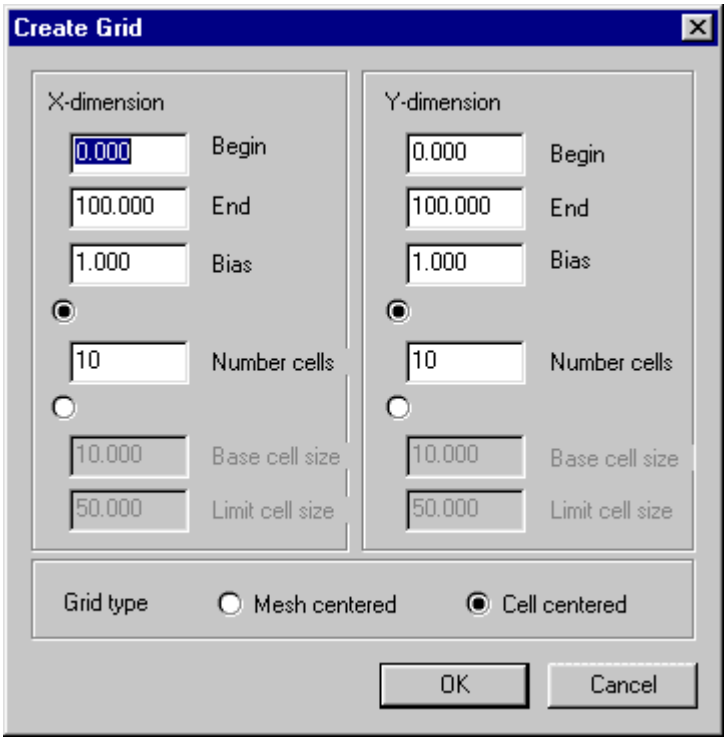


Figure 17-3 Create Grid Dialog.

Several options are available for defining the number and locations of the cell boundaries. A bias can be defined which controls how the cell size varies from one cell to the next. For example, an x bias of 1.5 will cause each cell to be 50% larger than the previous cell when moving in the positive x direction. The bias option should not be used when creating a grid to be used with *CASC2D* since that model requires a constant grid cell size.

The total number of cells in each direction (number of rows or columns) can be defined by explicitly entering a number or by entering a base cell size and a limit cell size. The base and limit cell size options are used when a bias other than 1.0 is specified. The base cell size is the size of the first cell in the sequence. The cells are then generated by altering the cell size according to the bias until the limit cell size is reached. The remainder of the cells are constructed using the limit cell size.

The controls at the bottom of the *Create Grid* dialog are used to define the type. The user can specify whether the grid should be a mesh-centered grid or a cell-centered grid.

17.6 Active / Inactive Cells

Each of the cells in a cell-centered grid can be active or inactive. An inactive cell is a cell which is not part of the computational domain. For example when

doing a surface runoff analysis using *CASC2D* cells inside the watershed boundary should be active while cells outside the boundary should be inactive.

An inactive cell is ignored when contours or fringes are displayed on the grid. A set of selected cells can be made inactive by selecting the *Inactivate Selected* command in the *Grid* menu. A set of inactive cells can be made active again by turning on the display of inactive cells using the *Display Options* dialog, selecting the cells, and selecting the *Activate Selected* command in the *Grid* menu (inactive cells can only be selected if they are being displayed).

17.6.1 Activate Polygon Region

In many cases it is useful to delineate the active/inactive regions in a grid using a polygon. A region can be activated by selecting the *Activate Polygon Region* from the *Grid* menu. A dialog appears prompting you to select either *Read polygon from file* or *Select polygon interactively*. If the *Read polygon from file* option is selected, *WMS* brings up the *File Browser* dialog, and prompts the user to specify a polygon file. If the *Select polygon interactively* option is selected, *WMS* prompts the user to define the polygon by selecting at least three points and double-clicking when done.

Once a polygon is entered, each cell is compared to the polygon. If the cell center is on the interior of the polygon, it is made active. Otherwise, the cell is made inactive.

17.7 Data Type Conversion

It is sometime useful to convert a 2D grid to a set of scattered data points or a TIN data structure. Data for the grid can then be used to perform operations available for scattered points or TINs.

17.7.1 Grid -> Scatter Points

The *Grid -> Scatter Points* command in the *Grid* menu is used to create a new scatter point set using the nodes or cells of a 2D grid. A copy is made of each of the data sets associated with the grid and the data sets are associated with the new scatter point set.

This command is useful for comparing the solutions from two separate simulations from different grids. For example, if two simulations have been performed with slightly different grids (base vs. plan) it may be useful to generate a contour or fringe plot showing the difference between the solutions. It is possible to generate a data set representing the difference between two data sets using the data calculator. However, the two data sets must be

associated with the same grid before the data calculator can be used. The data sets from one of the grids can be transferred to the other grid as follows:

1. Load the first grid and its data set into memory.
2. Convert the grid to a scatter point set using the *Grid -> Scatter Points* command.
3. Delete the first grid by selecting the *Delete All* command from the *Edit* menu.
4. Load the second grid and its data set into memory.
5. Switch to the *2D Scatter Point* module and select an interpolation scheme using the *Interpolation Options* command in the *Interpolation* menu.
6. Interpolate the data set to the second grid by selecting the *to 2D Grid* command from the *Interpolation* menu.

At this point, both data sets will be associated with the second grid and the data calculator can be used to compute the difference between the two data sets.

17.7.2 Grid -> TIN

A new TIN can be created from a 2D grid by selecting the *Grid -> TIN* command from the *Grid* menu. Two triangles are created from each cell in the grid. The active scalar data set becomes the z value of TIN vertices.

CHAPTER 18

2D Scatter Points

18.1 Introduction

The *2D Scatter Point* module is used to interpolate from groups of 2D scattered data points to *2D Grids*. They can also be used to represent rainfall gages (or radar locations for NEXRAD data) and then interpolated to the basin centroids of a TIN in order to develop a time-varying rainfall curve to be used in *HEC-1* or *TR-20*.

Interpolation is useful for setting up input data for analysis codes. For example, interpolation can be used to generate hydraulic conductivity of a 2D grid as input to a *CASC2D* simulation.

18.2 Scatter Point Sets

Each of the points from which values are interpolated are called scatter points. A group of scatter points is called a scatter point set. Each of the scatter points is defined by a set of xy coordinates.

Each scatter point set has a list of scalar data sets. Each data set represents a set of values that can be interpolated to a grid. When an interpolation command is selected, the active data set for the scatter point set is used in the interpolation process.

Multiple scatter point sets can exist at one time in memory. One of the scatter point sets is always designated as the "active" scatter point set. Interpolation is performed from the active scatter point set only. The active scatter point set

can be changed using the *Select Scatter Point Set* tool described below. Whenever a new scatter point set is read from a file or created, it becomes the active set.

18.3 Inputting Scatter Point Sets

Scatter point sets can be created by converting from other data types (TINs, or grids). For example, if a 2D grid is converted to a scatter point set, each of the nodes in the grid become a scatter point and each of the scalar data sets associated with the grid are copied to the data set list for the new scatter point set.

Scatter point sets can also be input from a text file. The file formats for scatter point sets are described in Chapter 20

18.4 Saving Scatter Point Sets

Scatter point sets may be saved to a text file using the *Save As* command in the *File* Menu. When a scatter point file is saved, you must specify in the *Save As* dialog whether the data sets will also be saved. If the data sets are saved as part of the super file then they can be read back in subsequent sessions using the *Open* command in *File* menu.

18.5 Tool Palette

The following tools are active in the dynamic portion of the *Tool Palette* whenever the *2D Scatter Point* module is active.

18.5.1 Select Scatter Point

The *Select Scatter Point* tool is used to select individual scatter points for displaying the coordinates and current function value of individual scatter points in the *Edit Window*.

18.5.2 Select Scatter Point Set

The *Select Scatter Point Set* tool is used to select entire scatter point sets for deletion or to designate the active scatter point set. When this tool is active, an icon appears at the centroid of the set for each of the scatter point sets. A scatter point set is selected by selecting the icon for the set.

A selected scatter point set can be made the active set by double clicking on the icon for the set or by selecting the *Make Active* command from the *Interpolate* menu.

18.6 Display Options

A scatter point set is displayed by drawing a symbol for each of the scatter points. The display options control the appearance of the symbol. The display options can be set by selecting the *Display Options* command in the *Display* menu and then selecting the *Scatter Point* tab.

The scatter point display options are as follows:The *Scatter Point Symbols* item is used to display a symbol at the location of each scatter point. The button to the left of the item is used to bring up a dialog listing the available symbols. The color of each of the scatter points in a set may be changed from this dialog also. The *Scatter Point Symbols* option applies to the active scatter point set only. To change the symbols for a scatter point set other than the active set, the set must first be made active by double clicking on the set with the *Select Scatter Point Set* tool or by selecting the set and selecting the *Make Active* command from the *Interpolation* menu prior to bringing up the *Display Options* dialog.

- The *Scatter Point Numbers* item is used to display the scatter point ID number next to each scatter point.

- The *Scatter Point Values* item is used to display the vertex value of the active data set next to each scatter point.

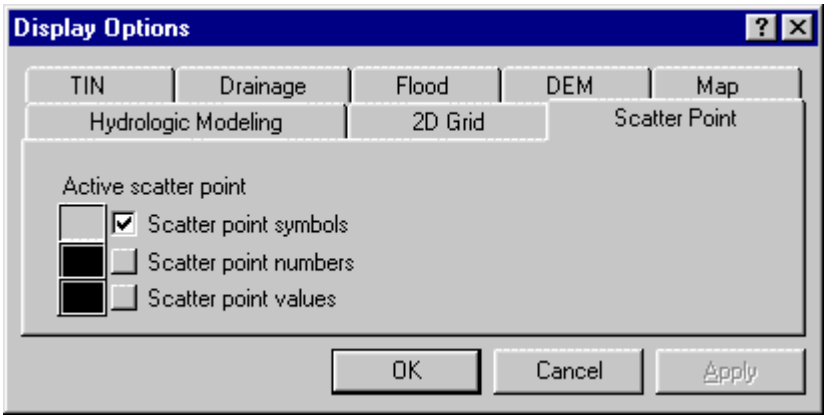


Figure 18-1 Scatter Point Display Options Dialog.

18.7 Interpolation Options

Scatter point sets are used for interpolation to other data types such as grids or basin centroids. Since no interpolation scheme is superior in all cases, several interpolation techniques are provided in *WMS*.

The interpolation option is selected using the *Interpolation Options* dialog accessed through the *Interpolation Options* command in the *Interpolation* menu (Figure 18-2). Once an option is selected, that option is used for all subsequent interpolation commands.

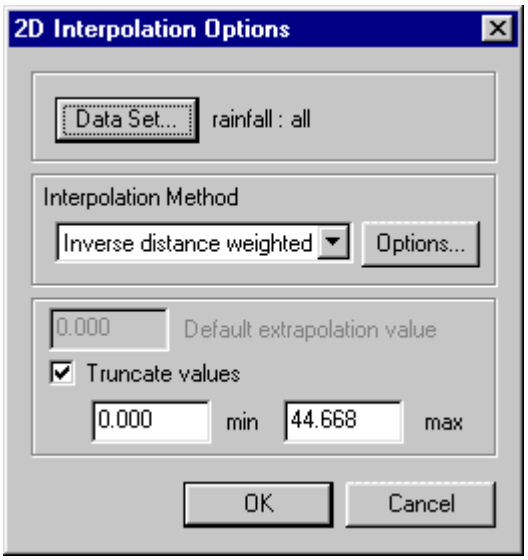


Figure 18-2 2D Interpolation Options Dialog.

Interpolation is always performed using the active scatter point set. By default the active data set and time step are interpolated. The active data set and time step can be selected using the *Data Browser* command in the *Data* menu or using the *Data Set.*. button at the top of the *Interpolation Options* dialog. This button also allows interpolation from all time steps of a transient data set to be performed.

When interpolating a set of values, it is sometimes useful to limit the interpolated values to lie between a minimum and maximum value. For example, when interpolating rainfall values, a negative value of rainfall is meaningless. However, many interpolation schemes will produce negative values even if all of the scatter points have positive data values. This occurs in areas where the trend in the data is toward a zero value. The interpolation may extend the trend beyond a zero value into the negative range. In such cases it is useful to limit the minimum interpolated value to zero. Interpolated values can be limited to a given range by selecting the Truncate values option in the *Interpolation Options* dialog and entering a minimum and maximum interpolation value.

The interpolation methods are listed in the *Interpolation Options* dialog. To the right of most of the method names is a button used to bring up a dialog for entering more interpolation options specific to the interpolation method. The methods supported for 2D interpolation are *linear, inverse distance weighted, Clough - Tocher,* and *natural neighbor*.

18.7.1 Linear Interpolation

If the *linear* interpolation scheme is selected, the scatter points are first triangulated to form a temporary TIN. If the surface is assumed to vary linearly across each triangle, the TIN describes a piece-wise linear surface which interpolates the scatter points. The equation of the plane defined by the three vertices of a triangle is as follows:

$$Ax + By + Cz + D = 0 \quad (18.1)$$

where A, B, and C, and D are computed from the coordinates of the three vertices (x_1,y_1,z_1), (x_2,y_2,z_2), & (x_3,y_3,z_3):

$$A = y_1(z_2 - z_3) + y_2(z_3 - z_1) + y_3(z_1 - z_2) \quad (18.2)$$

$$B = z_1(x_2 - x_3) + z_2(x_3 - x_1) + z_3(x_1 - x_2) \quad (18.3)$$

$$C = x_1(y_2 - y_3) + x_2(y_3 - y_1) + x_3(y_1 - y_2) \quad (18.4)$$

$$D = -Ax_1 - By_1 - Cz_1 \quad (18.5)$$

The plane equation can also be written as:

$$z = f(x,y) = -\frac{A}{C}\,x - \frac{B}{C}\,y - \frac{D}{C} \quad (18.6)$$

which is the form of the plane equation used to compute the elevation at any point on the triangle.

Since a TIN only covers the convex hull of a scatter point set, extrapolation beyond the convex hull is not possible with the linear interpolation scheme. Any points outside the convex hull of the scatter point set are assigned the default extrapolation value entered at the bottom of the *Interpolation Options* dialog.

18.7.2 Inverse Distance Weighted Interpolation

One of the most commonly used techniques for interpolation of scatter points is *inverse distance weighted (IDW)* interpolation. Inverse distance weighted methods are based on the assumption that the interpolating surface should be influenced most by the nearby points and less by the more distant points. The

interpolating surface is a weighted average of the scatter points and the weight assigned to each scatter point diminishes as the distance from the interpolation point to the scatter point increases. Several options are available for inverse distance weighted interpolation. The options are selected using the *Inverse Distance Weighted Interpolation Options* dialog (Figure 18-3).

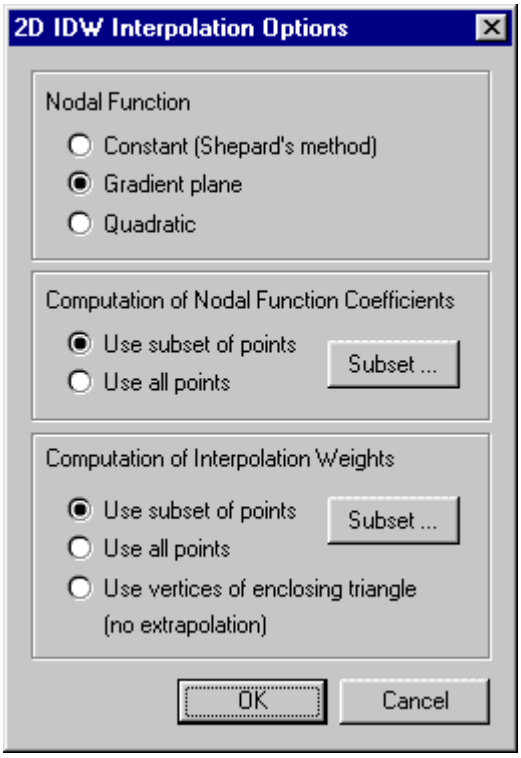


Figure 18-3 Inverse Distance Weighted Interpolation Options Dialog.

Shepard's Method

The simplest form of inverse distance weighted interpolation is sometimes called "Shepard's method" (Shepard 1968). The equation used is as follows:

$$F(x,y) = \sum_{i=1}^{n} w_i f_i \quad \text{.....(18.7)}$$

where n is the number of scatter points in the set , f_i are the prescribed function values at the scatter points (e.g. the data set values), and w_i are the weight functions assigned to each scatter point. The classical form of the weight function is:

$$w_i = \frac{h_i^{-p}}{\sum_{j=1}^{n} h_j^{-p}} \quad \text{.....(18.8)}$$

where p is an arbitrary positive real number called the power parameter (typically, p=2) and h_i is the distance from the scatter point to the interpolation point or

$$h_i=\sqrt{(x-x_i)^2+(y-y_i)^2} \quad \text{.....(18.9)}$$

where (x,y) are the coordinates of the interpolation point and (x_i,y_i) are the coordinates of each scatter point. The weight function varies from a value of unity at the scatter point to a value approaching zero as the distance from the scatter point increases. The weight functions are normalized so that the weights sum to unity.

The effect of the weight function is that the surface will interpolate each scatter point and be influenced most strongly between scatter points by the points closest to the point being interpolated.

Although equation 18.8 is typically used for the weight function in inverse distance weighted interpolation, the following equation is used in *WMS*:

$$w_i = \frac{\left[\frac{R-h_i}{Rh_i}\right]^2}{\sum_{j=1}^{n}\left[\frac{R-h_j}{Rh_j}\right]^2} \quad \text{.....(18.10)}$$

where h_i is the distance from the interpolation point to scatter point i, R is the distance from the interpolation point to the most distant scatter point, and n is the total number of scatter points. This equation has been found to give superior results to equation 18.8 (Franke & Nielson, 1980).

The weight function is a function of Euclidean distance and is radially symmetric about each scatter point. As a result, when interpolating elevation, the interpolating surface is somewhat symmetric about each point and tends toward the mean elevation of the scatter points between the scatter points. Shepard's method has been used extensively because it is very simple.

Gradient Planes Nodal Functions

A limitation of Shepard's method is that the interpolating surface is a simple weighted average of the data values of the scatter points and is constrained to lie between the extreme values in the data set. In other words, the surface will not infer local maxima or minima implicit in the data set. This problem can be overcome by generalizing the basic form of the equation for Shepard's method in the following manner:

$$F(x,y) = \sum_{i=1}^{n} w_i Q_i(x,y) \quad \text{.....(18.11)}$$

where Q_i are nodal functions or individual functions defined at each scatter point (Franke 1982; Watson & Philip 1985). The value of an interpolation point is calculated as the weighted average of the values of the nodal functions at that point. The standard form of Shepard's method can be thought of as a special case where horizontal planes (constants) are used for the nodal functions. The nodal functions can be sloping planes that pass through the scatter point. The equation for the plane is as follows:

$$Q_i(x,y) = f_x(x-x_i) + f_y(y-y_i) + f_i \quad \text{.....(18.12)}$$

where f_x and f_y are partial derivatives at the scatter point that have been previously estimated based on the geometry of the surrounding scatter points. Gradients are estimated in *WMS* by first triangulating the scatter points and computing the gradient at each scatter point as the average of the gradients of each of the triangles attached to the scatter point.

The planes represented by equation 18.12 are sometimes called "gradient planes." By averaging planes rather than constant values at each scatter point, the resulting surface infers extremities and is asymptotic to the gradient plane at the scatter point rather than forming a flat plateau at the scatter point.

Quadratic Nodal Functions

The nodal functions used in inverse distance weighted interpolation can also be higher degree polynomial functions constrained to pass through the scatter point and approximate the nearby points in a least squares manner. Quadratic polynomials have been found to work very well (Franke & Nielson 1980; Franke 1982). The resulting surface reproduces local variations implicit in the data set, is very smooth, and will approximate the quadratic nodal functions near the scatter points. The equation used for the quadratic nodal function centered at point k is as follows:

$$Q_k(x,y) = a_{k1}+a_{k2}(x-x_k)+a_{k3}(y-y_k)+a_{k4}(x-x_k)^2$$

$$+a_{k5}(x-x_k)(y-y_k)+a_{k6}(y-y_k)^2 \quad \text{.....(18.13)}$$

To define the function, the six coefficients $a_{k1}..a_{k6}$ must be found. Since the function is centered at the point k and passes through point k, we know beforehand that $a_{k1}=f_k$ where f_k is the function value or z-value at point k. The equation simplifies to:

$$Q_k(x,y) = f_k+a_{k2}(x-x_k)+a_{k3}(y-y_k)+a_{k4}(x-x_k)^2$$

$$+a_{k5}(x-x_k)(y-y_k)+a_{k6}(y-y_k)^2 \quad (18.14)$$

Now there are only five unknown coefficients. The coefficients are found by fitting the quadratic to the nearest N_Q scatter points in a weighted least squares fashion. In order for the matrix equation used to solve for the coefficients to be stable, there should be at least five scatter points in the set.

Interpolation Subsets

In the *IDW Interpolation Options* dialog shown in Figure 18-4, an option is available for using a subset of the scatter points (as opposed to all of the available scatter points) in the computation of the nodal function coefficients and in the computation of the interpolation weights. Using a subset of the scatter points drops distant points from consideration since they are unlikely to have a large influence on the nodal function or on the interpolation weights. In addition, using a subset can speed up the computations since less points are involved.

If the *Use subset of points* option is chosen, the *Subsets* button can be used to bring up the *Subset Definition* dialog shown in Figure 18-4. Two options are available for defining which points are included in the subset. In one case, only the nearest N points are used. In the other case, only the nearest N pts in each quadrant are used. This approach may give better results if the scatter points tend to be clustered.

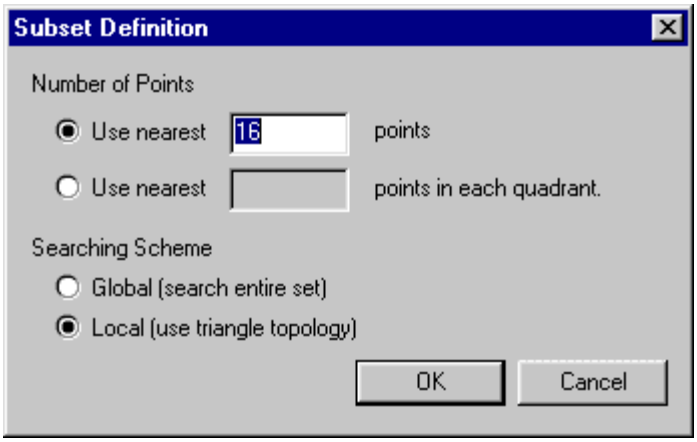


Figure 18-4 Subset Definition Dialog.

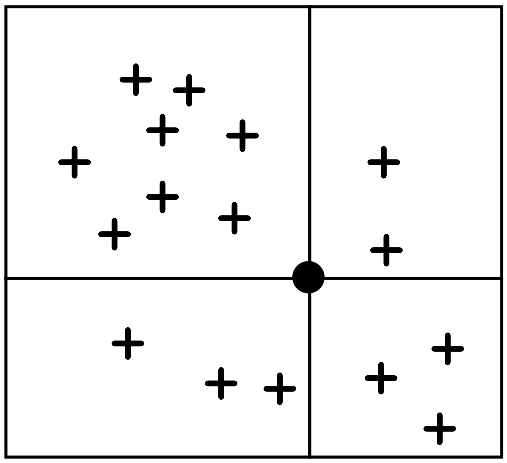


Figure 18-5 The Four Quadrants Surrounding an Interpolation Point.

If a subset of the scatter point set is being used for interpolation, a scheme must be used to find the nearest N points. Two methods for finding a subset are provided in the *Subset Definition* dialog: the global method and the local method. With the global method, each of the scatter points in the set are searched for each interpolation point to determine which N points are nearest the interpolation point. This technique is fast for small scatter point sets but may be slow for large sets.

With the local scheme, the scatter points are triangulated to form a temporary TIN before the interpolation process begins. To compute the nearest N points, the triangle containing the interpolation point is found and the triangle topology is then used to sweep out from the interpolation point in a systematic fashion until the N nearest points are found. The local scheme is typically much faster than the global scheme for large scatter point sets.

Local Weighting Method

As mentioned above, it is possible to localize the search for the nearest N scatter points to the interpolation point using the topology of a TIN constructed from the scatter points. Yet another scheme is available for making the interpolation process a local scheme by taking advantage of TIN topology (Franke & Nielson, 1980). With this technique, the subset of points used for interpolation consists of the three vertices of the triangle containing the interpolation point. The weight function or blending function assigned to each scatter point is an cubic S-shaped function (Figure 18-6a). The fact that the slope of the weight function tends to unity at its limits ensures that the slope of the interpolating surface will be continuous across triangle boundaries.

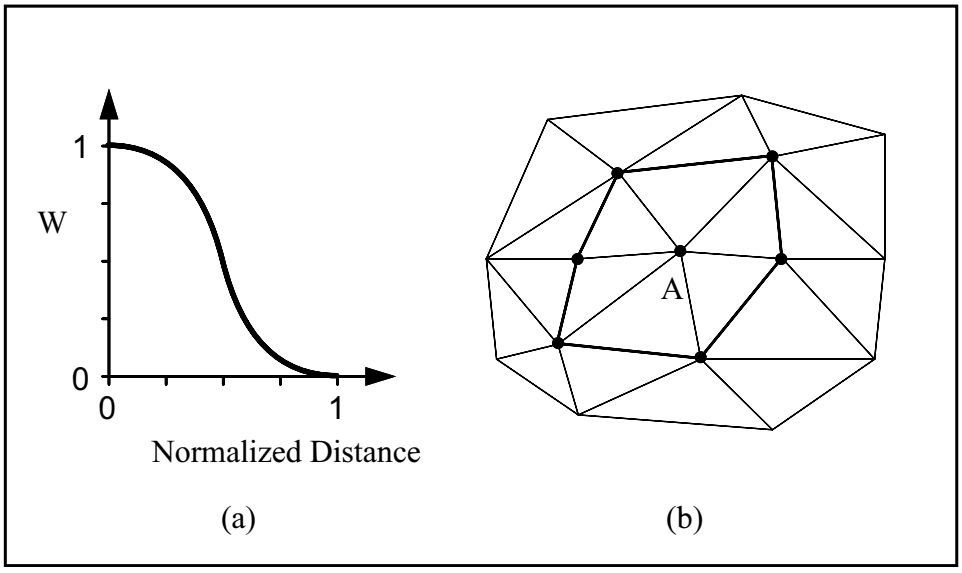


Figure 18-6 (a) S-Shaped Weight Function and (b) Delauney Point Group for Point A.

The influence of the weight function extends over the limits of the Delauney point group of the scatter point. The Delauney point group is the "natural neighbors" of the scatter point, and the perimeter of the group is made up of the outer edges of the triangles that are connected to the scatter point as shown in Figure 18-6b. The weight function varies from a weight of unity at the scatter point to zero at the perimeter of the group. For every interpolation point in the interior of a triangle there are three nonzero weight functions (the weight functions of the three vertices of the triangle). For a triangle T with vertices i, j, & k, the weights for each vertex are determined as follows:

$$W_i(x,y) = b_i^2(3 - 2b_i)$$

$$+ 3\frac{b_i^2 b_j b_k}{b_i b_j + b_i b_k + b_j b_k}\left\{b_j\left[\frac{\|e_i\|^2+\|e_k\|^2-\|e_j\|^2}{\|e_k\|^2}\right]+b_k\left[\frac{\|e_i\|^2+\|e_j\|^2-\|e_k\|^2}{\|e_j\|^2}\right]\right\} \quad ..(18.15)$$

where $\|e_i\|$ is the length of the edge opposite vertex i, and b_i, b_j, b_k are the area coordinates of the point (x,y) with respect to triangle T. Area coordinates are coordinates that describe the position of a point within the interior of a triangle relative to the vertices of the triangle. The coordinates are based solely on the geometry of the triangle. Area coordinates are sometimes called "barycentric coordinates." The relative magnitude of the coordinates corresponds to area ratios as shown in Figure 18-7.

The xy coordinates of the interior point can be written in terms of the xy coordinates of the vertices using the area coordinates as follows:

$$x = b_i x_i + b_j x_j + b_k x_k \quad .....(18.16)$$

$$y = b_i y_i + b_j y_j + b_k y_k \quad (18.17)$$

$$1.0 = b_i + b_j + b_k \quad (18.18)$$

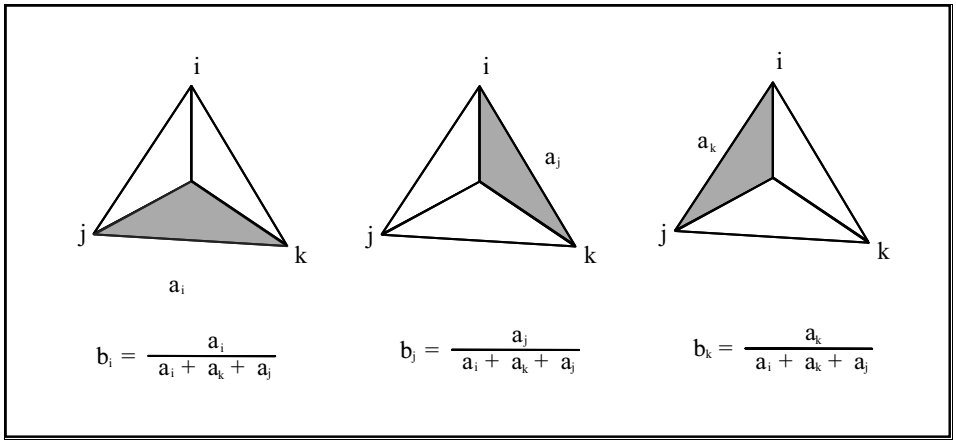


Figure 18-7 Barycentric Coordinates for a Point in a Triangle.

Solving the above equations for b_i, b_j, and b_k yields:

$$b_i = \frac{1}{2A}[(x_j y_k - x_k y_j) + (y_j - y_k)x + (x_k - x_j)y] \quad (18.19)$$

$$b_j = \frac{1}{2A}[(x_k y_i - x_i y_k) + (y_k - y_i)x + (x_i - x_k)y] \quad (18.20)$$

$$b_k = \frac{1}{2A}[(x_i y_j - x_j y_i) + (y_i - y_j)x + (x_j - x_i)y] \quad (18.21)$$

$$A = \frac{1}{2}(x_i y_j + x_j y_k + x_k y_i - y_i x_j - y_j x_k - y_k x_i) \quad (18.22)$$

Using the weight functions defined above, the interpolating surface at points inside a triangle is computed as:

$$F(x,y) = W_i(x,y)Q_i(x,y) + W_j(x,y)Q_j(x,y) + W_k(x,y)Q_k(x,y) \quad (18.23)$$

where W_i, W_j, and W_k are the weight functions and Q_i, Q_j, and Q_k are the nodal functions for the three vertices of the triangle.

IDW interpolation using local weights as defined in equation 18.15 can be selected in the *Weighting Method* section of the *IDW Interpolation Options* dialog. This type of weighting is significantly faster than the standard weighting method, particularly if the number of scatter points is large.

18.7.3 Clough - Tocher Interpolation

The *Clough-Tocher* interpolation technique is often referred to in the literature as a finite element method because it has origins in the finite element method of numerical analysis. Before any points are interpolated, the scatter points are first triangulated to form a temporary TIN. A bivariate polynomial is defined over each triangle, creating a surface made up of a series of triangular Clough-Tocher surface patches.

The Clough-Tocher patch is a cubic polynomial defined by twelve parameters shown in Figure 18-8: the function values, f, and the first derivatives, f_x & f_y, at each vertex, and the normal derivatives, $\partial f/\partial n$, at the midpoint of the three edges in the triangle (Clough & Tocher, 1965; Lancaster & Salkauskas, 1986). The first derivatives at the vertices are estimated using the average slopes of the surrounding triangles. The element is partitioned into three subelements along seams defined by the centroid and the vertices of the triangle.

A complete cubic polynomial of the form:

$$F(x,y)=\sum_{j=0}^{3-i} c_{ij}x^i y^j \quad \text{.....(18.24)}$$

is created over each subtriangle with slope continuity across the seams and across the boundaries of the triangle. Second derivative continuity is not maintained across the seams of the triangle.

The form of equation 18.24 implemented in *WMS* is highly complex and is not included in this reference manual. The complete set of equations can be found in Jones (1990).

Since the Clough-Tocher scheme is a local scheme, it has the advantage of speed. Even very large scatter point sets can be interpolated very quickly. It also tends to give a very smooth interpolating surface which brings out local trends in the data set quite accurately.

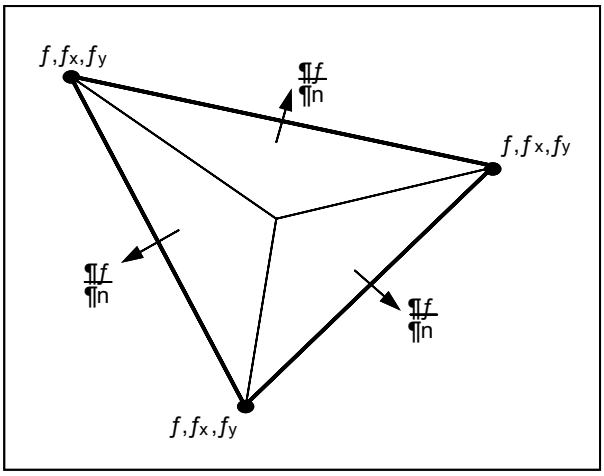


Figure 18-8 The Twelve Parameters Used to Define the Clough-Tocher Triangle.

Since a TIN only covers the convex hull of a scatter point set, extrapolation beyond the convex hull is not possible with the Clough-Tocher interpolation scheme. Any points outside the convex hull of the scatter point set are assigned the *default extrapolation value* entered at the bottom of the *Interpolation Options* dialog.

18.7.4 Natural Neighbor Interpolation

Natural neighbor interpolation is also supported in *WMS*. Natural neighbor interpolation has many positive features. It can be used for both interpolation and extrapolation, and it behaves very well with clustered scatter points. Natural neighbor interpolation was first introduced by Sibson (1981). A more detailed description of natural neighbor interpolation in multiple dimensions can be found in Owen (1992).

The basic equation used in natural neighbor interpolation is identical to the one used in IDW interpolation (equation 18.11). As with IDW interpolation, the nodal functions can be either constants, gradient planes, or quadratics. The nodal function can be selected using the *Natural Neighbor Interpolation Options* dialog (Figure 18-9). The difference between IDW interpolation and natural neighbor interpolation is the method used to compute the weights and the method used to select the subset of scatter points used for interpolation.

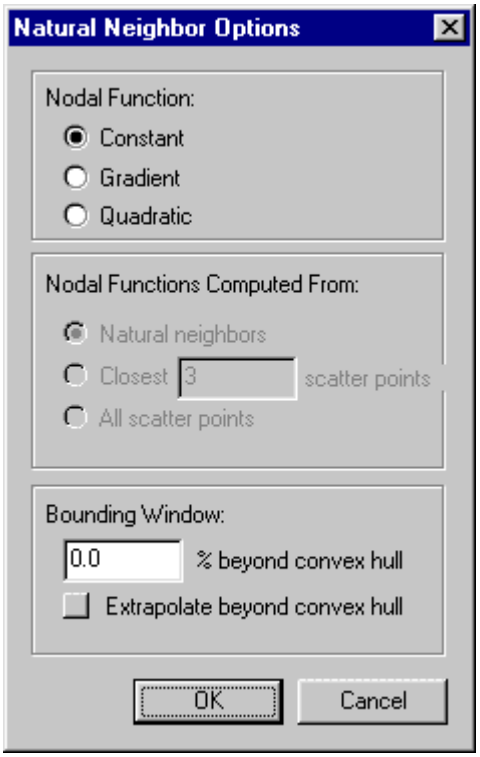


Figure 18-9 Natural Neighbor Options Dialog.

Natural neighbor interpolation is based on the Thiessen polygon network of the scatter point set. The Thiessen polygon network can be constructed from the Delauney triangulation of a scatter point set (Figure 18-10). A Delauney triangulation is a TIN that has been constructed so that the Delauney criterion has been satisfied (see section 6.6 on triangulation).

There is one Thiessen polygon in the network for each scatter point. The polygon encloses all area that is closer to the enclosed scatter point than any other scatter point. The polygons in the interior of the scatter point set are closed polygons and the polygons on the convex hull of the set are open polygons.

Each Thiessen polygon is constructed using the circumcircles of the triangles resulting from a Delauney triangulation of the scatter points. The vertices of the Thiessen polygons correspond to the centroids of the circumcircles of the triangles.

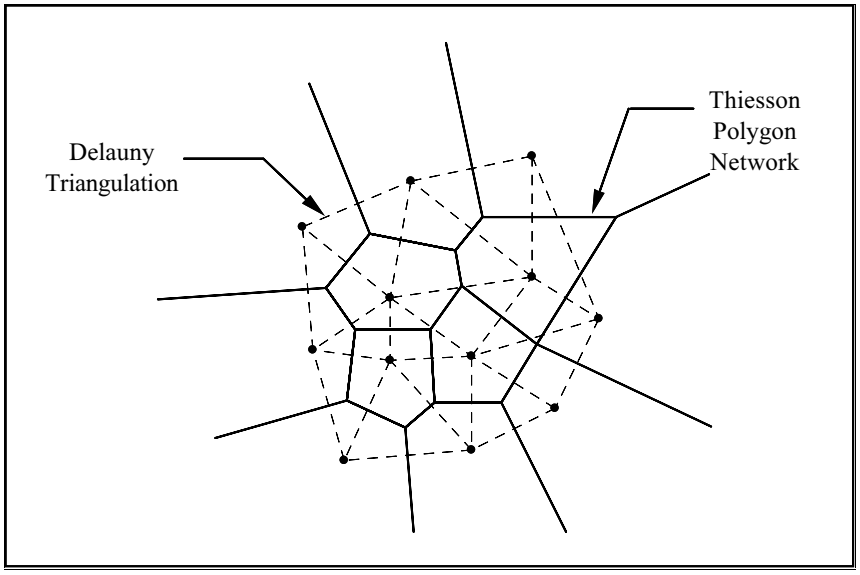


Figure 18-10 Delauney Triangulation and Corresponding Thiessen Polygon Network for a Set of Scatter Points.

Local Coordinates

The weights used in natural neighbor interpolation are based on the concept of local coordinates. Local coordinates define the "neighborliness" or amount of influence any scatter point will have on the computed value at the interpolation point. This neighborliness is entirely dependent on the area of influence or Thiessen polygons of the surrounding scatter points.

To define the local coordinates for the interpolation point, P_n, the area of all Thiessen polygons in the network must be known. Temporarily inserting P_n into the TIN will cause the TIN and the corresponding Thiessen network to change, resulting in new Thiessen areas for the polygons in the neighborhood of P_n.

The concept of local coordinates is shown graphically in Figure 18-11. Points 1-10 are scatter points and P_n is a point where some value associated with points 1-10 is to be interpolated. The dashed lines show the edges of the Thiessen network before P_n is temporarily inserted into the TIN and the solid lines show the edges of the Thiessen network after P_n is inserted.

Only those scatter points whose Thiessen polygons have been altered by the temporary insertion of P_n are included in the subset of scatter points used to interpolate a value at P_n. In this case, only points 1, 4, 5, 6, & 9 are used. The local coordinate for each of these points with respect to P_n is defined as the area shared by the Thiessen polygon defined by point P_n and the Thiessen polygon defined by each point before point P_n is added. The greater the

common area, the larger the resulting local coordinate, and the larger the influence or weight the scatter point has on the interpolated value at P_n.

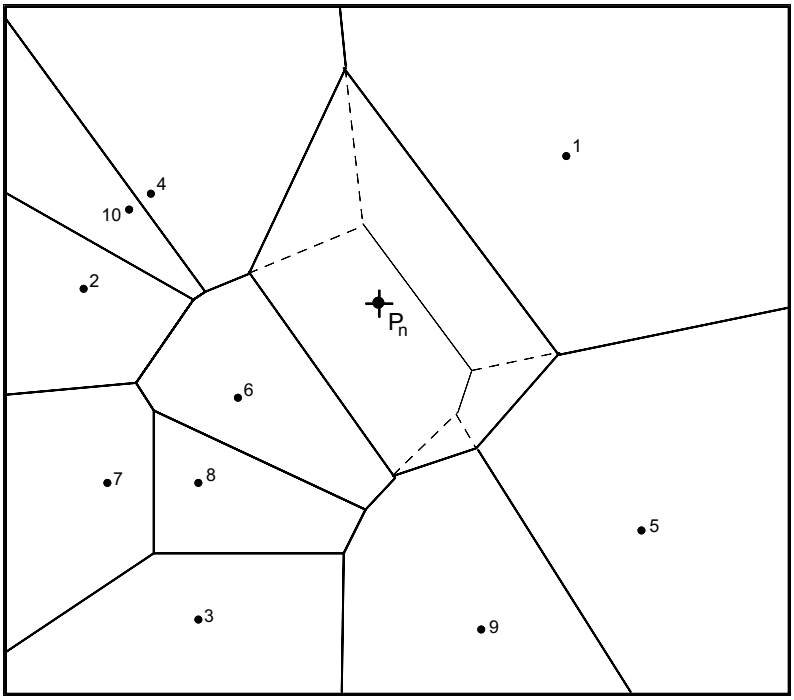


Figure 18-11 Overlapping Thiessen Polygon Areas Used in Computation of Local Coordinates.

If we define $\kappa(n)$ as the Thiessen polygon area of P_n and $\kappa_m(n)$ as the difference in the Thiessen polygon area of a neighboring scatter point, P_m, before and after P_n is inserted, then the local coordinate $\lambda_m(n)$ is defined as:

$$\lambda_m(n) = \frac{\kappa_m(n)}{\kappa(n)} \quad \text{.....(18.25)}$$

The local coordinate $\lambda_m(n)$ varies between zero and unity. If P_n is at precisely the same location as P_m, then the Thiessen polygon areas for P_n and P_m are identical and $\lambda_m(n)$ has a value of unity. In general, the greater the relative distance P_m is from P_n, the smaller its influence on the final interpolated value.

The weights used in natural neighbor interpolation are computed by normalizing the local coordinates so that they sum to one:

$$w_m(n) = \frac{\lambda_m(n)}{\sum_{i=1}^{p} \lambda_i(n)} \quad \text{.....(18.26)}$$

where $w_m(n)$ is the weight of scatter point P_m with respect to the interpolation point P_n, and p is the number of points in the neighborhood of P_n with non-zero local coordinates.

Bounding Window

As shown in Figure 18-10, the Thiessen polygons for scatter points on the perimeter of the TIN are open-ended polygons. Since such polygons have an infinite area, they cannot be used directly for natural neighbor interpolation. In order to make the area of these polygons finite, a bounding box or window is superimposed on the scatter point set (Figure 18-12). Polygons for points on the exterior of the scatter point set are clipped or truncated to the bounding window.

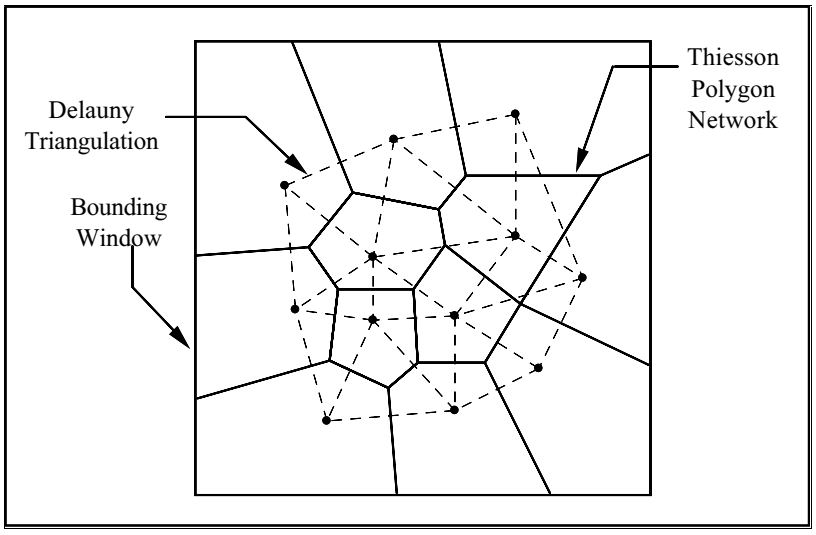


Figure 18-12 Bounding Window Used in Natural Neighbor Interpolation.

With finite Thiessen polygon areas on the perimeter of the scatter point set it is possible to perform extrapolation (estimate values for interpolation points outside the convex hull of the scatter point set) as well as interpolation. However, the value computed by extrapolation are somewhat influenced by the relative size of the bounding window with respect to the size of the scatter point set. The larger the bounding window, the greater the influence of the perimeter scatter points on the extrapolated values. If the bounding window is extremely large, the extrapolated values will be influenced only by the points on the convex hull of the scatter point set. If the bounding window is not significantly larger than the scatter point set, the extrapolated values will be

influenced by interior scatter points in the neighborhood of the interpolation point in addition to perimeter scatter points near the interpolation point. The relative size of the bounding window can be altered using the *Natural Neighbor Interpolation* Options dialog. In addition, the dialog can be used to turn off the extrapolation option entirely.

18.8 Interpolation To Grids

Once an interpolation scheme has been selected and all of the parameters for the selected scheme have been input, the data associated with the active time step and data set of the active scatter point set can be interpolated to a grid. During the interpolation process, a new data set is constructed for the grid containing the interpolated values.

The interpolation is done either to the grid nodes or to the grid cell centers depending on whether the grid is a mesh or cell centered grid.

18.9 Interpolating Rainfall To Basin Centroids

The *Interpolate to Basin Precip* command is designed to interpolate rainfall values at scattered points to the xy series representing rainfall for a basin in either *HEC-1* or *TR-20*. The scattered points typically represent either gaging stations or radar locations for NEXRAD data. Unlike interpolation to grids, this command does not use the active interpolation method, but rather uses the Thiessen method in order to assign the weights of each scatter point for each basin.

In addition to interpolating rainfall values to basin centroids it is often convenient to interpolate the rainfall values to a grid so that an animation sequence of a storm can be generated. The Bounding Grid options described below is useful for setting up a grid for this purpose.

18.10 Bounding Grid

In many cases, it is useful to interpolate to a grid which just contains the scatter point set where the data are defined. The *Bounding Grid* command was designed in order to simplify the creation of such a grid. If the *Bounding Grid* command in the *Data* menu is selected, the *Create Grid* dialog comes up with the grid dimensions automatically initialized so that the grid extends beyond the bounds of the active scatter point set by ten per cent.

CHAPTER 19

CASC2D Interface

19.1 Introduction

CASC2D is a physically based rainfall/runoff model which uses rectangular grid cells to represent the distributed watershed and rainfall domains. The model uses a two-dimensional diffusive wave equation to simulate overland flow and a one-dimensional diffusive wave equation to simulate channel flow. It includes an advanced soil moisture accounting procedure, primarily based on the Green-Ampt infiltration model. *WMS* can be used to create a finite difference grid and provide all necessary input, such as stream locations and properties, lakes, soil properties and precipitation. Spatial data can also be prepared using standard GIS software and then imported into *WMS*.

19.2 Typical Steps To Set Up A Casc2d Model

Because of the variety of tools available in *WMS*, there are a lot of different ways in which a *CASC2D* model can be created. However, the following outlines the typical steps which must be completed in order to build a *CASC2D* model:

1. Create a Grid - A grid is usually created in one of three different ways:

a) Using the *Create Grid* command from the *Grid* menu, and then manually specifying which cells are active (inside the watershed boundary) and inactive (outside the watershed boundary). Grid cell elevations must be interpolated from a scattered data set when manually creating the grid.

b) Using a boundary feature polygon and the *Create Grid* command in the *Feature Objects* menu. With this option you should use either a background TIN or DEM to interpolate elevations for the newly created grid. Active and inactive cells are automatically assigned according to the boundary feature polygon. When creating *CASC2D* models with feature objects the coverage type should be set to *CASC2D* using the *Coverages* dialog found in the *Feature Objects* menu.

c) Importing an *ARC/INFO*® or GRASS grid using the *ARC/INFO or GRASS grid to 2D grid* option found in the *Import* dialog in the *File* menu. The values in the imported grid are used to define elevations.

2. Initialize *CASC2D* data and setup up job control parameters.

3. Assign overland flow parameters.

4. Assign infiltration parameters.

5. Define the rainfall values.

6. Define stream channels and cross-section properties.

7. Run the *Model Checker*.

8. Save the project file.

9. Run *CASC2D*.

19.3 Tool Palette

The majority of the *CASC2D* model is set up from with in the *Grid* module so the required tools are the same as for that module. Stream channels are set up using arcs in the *Map* module.

19.4 Reading And Writing Casc2d Project Files

Several different files are required to run a *CASC2D* simulation. In most cases there is a separate grid file for each grid cell attribute, along with files for precipitation and channel routing. In order to conveniently manage all of these files *WMS* uses a project file. The project file is a file which contains the options and names of files being used for a particular simulation.

Whenever a *CASC2D* project is saved from *WMS* all of the currently defined options and filenames are saved to a project file. In addition, all options which require the use of a separate file are saved. When reading a project file the

options and all accompanying files are read back into *WMS*. Furthermore, the *CASC2D* program distributed with *WMS* reads this same project file so that it can be conveniently executed using the project file name as a single command line argument.

While the project file itself could be edited it is important not to change the order or delete some of the options manually since certain aspects of a *CASC2D* simulation must be created ahead of others. Altering the order of the options in the project file could cause problems the next time the project file is read into *WMS* or used with *CASC2D*.

Because there are so many files involved in a *CASC2D* project, it is a good idea to create a separate directory for each project. This will help avoid problems associated with mixing files of different simulations together.

19.4.1 Saving a CASC2D Project

The *Save Project File* command is used to save *CASC2D* project files. After issuing this command a dialog appears with numerous file names (Figure 19-1). File names for options which are active are undimmed while those for options which are not currently being used are dimmed. While this dialog may seem foreboding, it is actually very simple to use. The name of the project file is shown at the top of the dialog, and can be changed by clicking on the *File Browser* button. Once the project file name is specified the name in the prefix edit field is updated to reflect the prefix of project file. This same prefix can then be applied to all files by clicking on the *Update* button to the right of the edit field. The other file name edit fields need not be changed manually, but are listed separately so that single files can be renamed with other prefixes if the need arises.

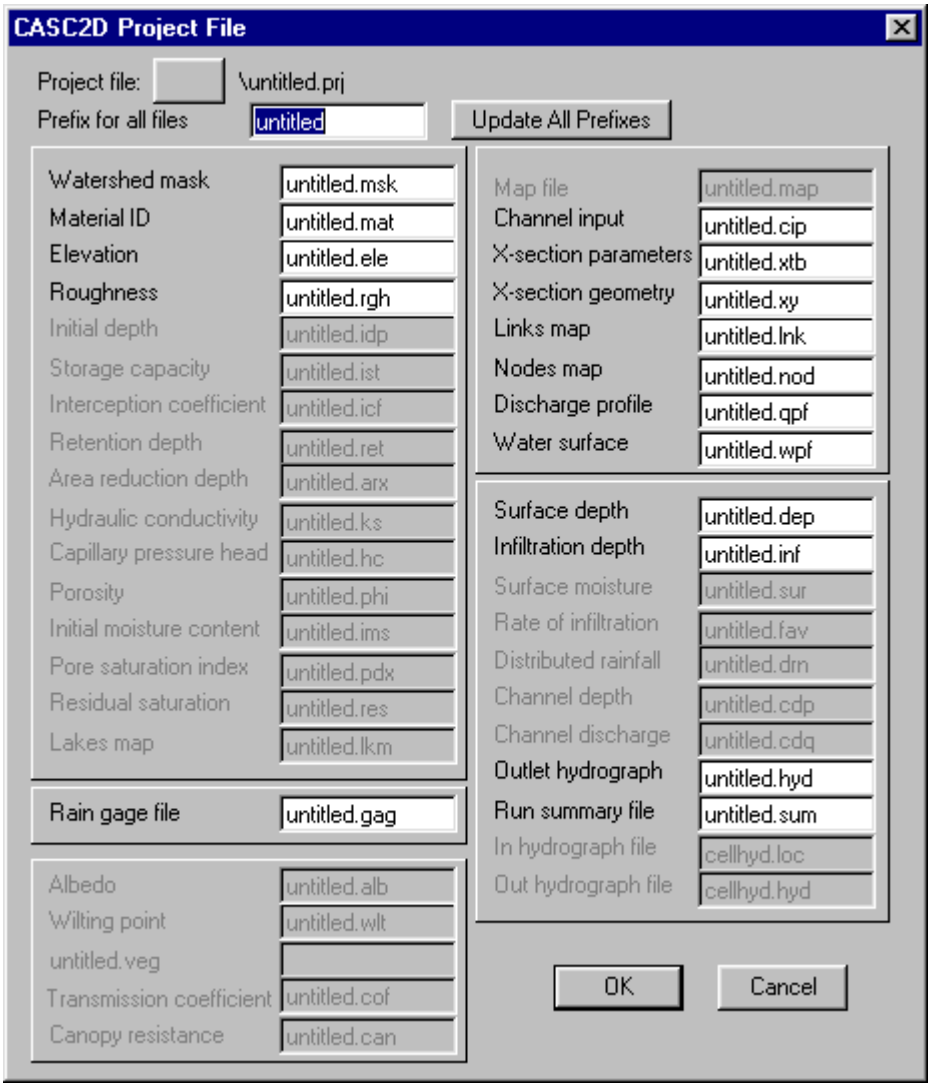


Figure 19-1 CASC2D Save Project Dialog.

19.4.2 Reading a CASC2D Project

The *Read Project File* command is used to read in all options and accompanying files for a *CASC2D* project.

19.5 Creating A Grid

The first task in defining a *CASC2D* model is to create a finite difference grid, with accompanying elevations for the domain of the watershed being analyzed. General methods for grid creation are discussed in the *Grid* module chapter, but three different techniques are reviewed here since they are the most common ways grids are created for *CASC2D* models.

19.5.1 Manually Creating a Grid

The simplest way to set up a grid is to use the *Create Grid* command from the *Grid* menu of the *Grid* module. However, while this allows a grid to quickly be generated, it leaves two important issues unresolved: elevations for the grid cells, and which cells within the rectangular grid are inside or outside the watershed boundary. If a grid is created manually then elevations must be interpolated from a scattered data set, or imported from a grid or data set of the exact same size from the elevations map (accessible from the *Overland Flow* options in the *CASC2D* menu.

By default all cells of a grid are active or inside the watershed. Cells outside the watershed boundary can be made inactive by using any or combinations of cell selecting techniques, and then choosing the *Inactivate Selected* option from the *Grid* menu.

19.5.2 Creating a Grid from Feature Objects

Most of the time elevations for a *CASC2D* model will be derived from DEMs. The tools in the DEM module of *WMS* can be used to import and stitch together adjacent USGS DEM files, or import GIS files for use as background elevation maps. In addition a set of scattered elevation points could be read in and triangulated into a TIN for use as a background elevation map. Once an elevation map has been set up, a boundary feature polygon can be used in conjunction with the *Create Grid* command in the *Feature Objects* menu of the *Map* module to generate a grid within the defined boundary, using the background elevation map to define grid cell elevations.

The boundary feature polygon should not extend beyond the limits of the background elevation map. When the *Create Grid* command (from the *Feature Objects* menu) is issued the standard *Create Grid* dialog box comes up with dimensions that just bound the feature polygon. The grid size can be specified and when the grid is created, cells outside the boundary feature polygon are automatically inactivated.

The boundary feature polygon can be created using any one of the different options available in *WMS*. For example, a rough boundary could be created in the *Map* module using contours of the background elevation map as a guide. This may require that a "fine tuning" of active and inactive grid cells be done manually after the grid is created. Further, the rough boundary could be used to create a TIN first and then a watershed delineation performed using the tools of the *TIN* module. Once a more precise watershed boundary is defined using a TIN, the TIN boundary can be converted to a boundary feature polygon and used to create the grid.

19.5.3 Importing a GIS Based Grid

Typical GIS software has tools for creating grids, and delineating watersheds using a grid based data structure. *WMS* can import grids from both the *ARC/INFO*® and GRASS GIS's. The grid cell values should be either on or off (1 or zero) representing cells that are inside (1) and outside (0) the watershed boundary. In order to establish elevations for the grid cells, a separate grid file from the GIS should be exported containing the elevations. This grid can then be imported (as explained in the section below on editing map parameters) from the *Edit Map* dialogs that comes up from the *Overland Flow Properties* command.

19.6 Job Control Parameters

The *Job Control* command allows all general run time options for a *CASC2D* model to be defined. These options are saved as part of the project file. The dialog used to define these parameters is shown in Figure 19-2, and definitions follow.

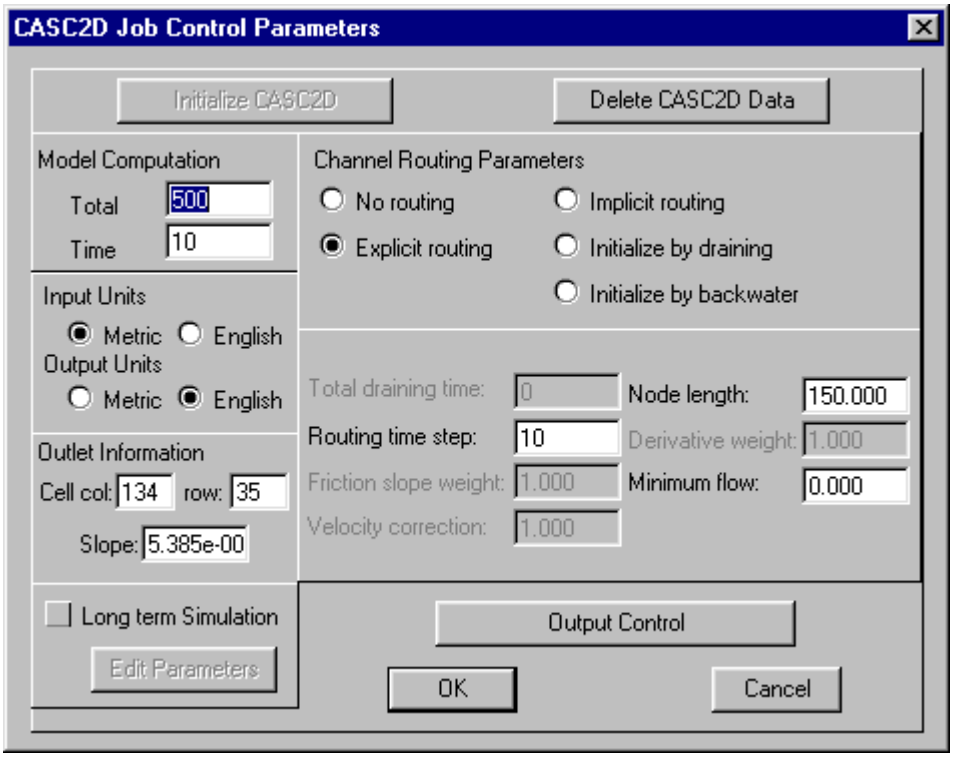


Figure 19-2 CASC2D Job Control Dialog.

19.6.1 Initializing CASC2D Data

Because of the numerous parameters available for each grid cell, the amount of memory for a large grid can become large. For this reason, and because grids have multiple uses within *WMS*, computer memory for *CASC2D* parameters is not created until a *CASC2D* simulation is initialized. This is done by selecting the *Initialize CASC2D* button at the top of the Job Control dialog. The *Delete CASC2D Data* button can be used to free up the memory required for setting up *CASC2D* parameters. Until *CASC2D* data is initialized, all other options and menus are dimmed.

19.6.2 Computation Time

The total time and time step for the model computation can be specified from this dialog. The total time should be specified in minutes and the time step in seconds.

19.6.3 Units

The units flag is used to specify whether the grid size and elevations are in metric or SI units. *CASC2D* requires that all data be entered in metric units. The one exception is the grid origin and cell size values, and the elevations. If the English units toggle box is checked the grid size and elevation values are converted from feet to meters. No scaling is done for the other input parameters of *CASC2D*.

19.6.4 Outlet Cell Information

In the absence of a stream channel the outlet grid cell must be identified by it's I and J index so that a runoff hydrograph can be generated at the proper location. When a channel is present the outlet cell information is not required since the outlet cell is implicit from the channel network.

The slope at the outlet is the representative slope leading from the outlet off the model.

19.6.5 Channel Routing Parameters

A *CASC2D* model can be run with or without the definition of stream channels. Without them only the surface runoff computations are performed. This is always the default option when initializing a *CASC2D* project. However, coupled hydraulic routing can be done using the Priessman option. The other two options for channel routing exist only to initialize channel flows before actually running the runoff model. These two methods include initialize by draining and initialize by backwater. When either of the latter two options are specified, execution of *CASC2D* will only cause channel flow

files (depth, and discharge) to be initialized. Once the channels have been successfully initialized, the channel option should be switched to Priessman routing for a full analysis to be done.

The total routing time, routing time step, and minimum flow values apply to the initialization of channels. The total routing time is specified in minutes and the routing time step in seconds. They are used, along with the minimum flow value to determine convergence for the initialization problem.

The velocity correction factor, friction slope weight and spatial derivative weight are used to control the stability and speed of channel routing during a *CASC2D* analysis. A more in depth discussion of these parameters is provided in the *CASC2D* primer.

19.6.6 Output Control

The output control button brings up a dialog which lets the user specify which output options are active for a given project. *CASC2D* can generate ASCII or binary data sets, or GRASS grid files for distributed rainfall intensity, surface depth, cumulative infiltration depth, surface soil moisture, infiltration rate, channel cell depth and channel cell discharge. By turning in one of these options on a file is created with the specified output at the time step interval specified by the write step edit field. For example if surface depth is turned on with a write step of 10, then values of surface depth for every cell are written to a file every tenth time step. The file name is specified in the *Save Project* dialog, and by default will contain the same prefix as the project file name.

CASC2D always generates as part of the output a discharge hydrograph for the outlet. The hydrograph write step is used to determine how many time steps should elapse between consecutive values (ordinates) of the hydrograph.

The suppress printing toggle controls whether or not output to the screen is displayed during execution of *CASC2D*.

19.7 Precipitation

Two different options for defining precipitation for *CASC2D* models are available from the *WMS* interface. The first is uniform rainfall over the entire watershed and the second allows gage data at specified locations to be defined.

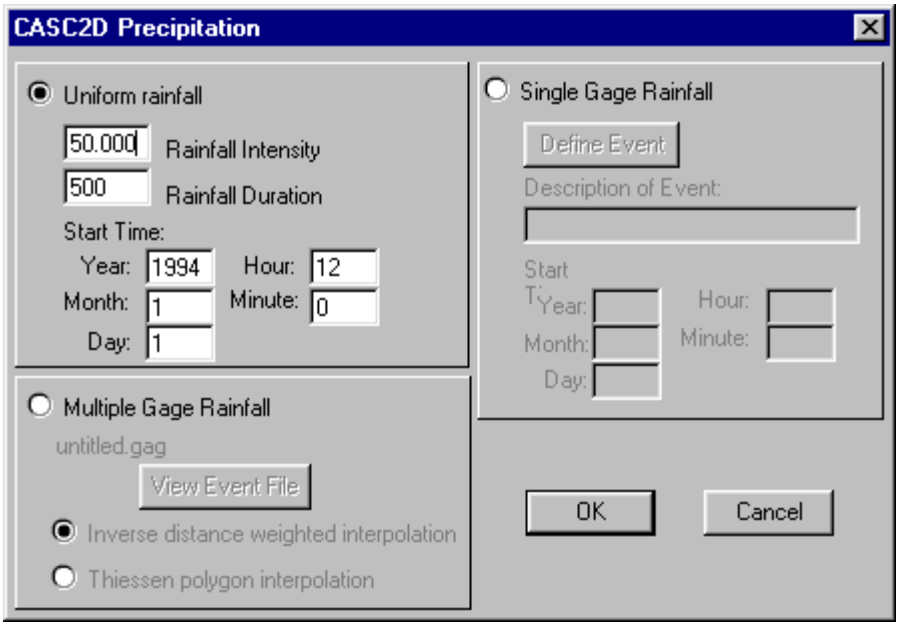


Figure 19-3 CASC2D Precipitation Dialog.

19.7.1 Uniform Rainfall

The *Uniform Rainfall* option requires that a single rainfall intensity curve for the entire watershed to be defined. This is done by bringing up the *XY-Series Editor* be clicking on the *Define Storm* button. Rainfall intensity values are defined for the given intervals. Time intervals can be changed by bringing up the *XY Opts* dialog from within the *XY-Series Editor*.

19.7.2 Gages

Rainfall intensity curves can be defined for any number of gage locations. Each gage must be defined by specifying the name, x-y location, and a time-varying rainfall intensity curve. Gages are created/edited by selecting the *Define Gages* button. This dialogs allows new gages to be created, existing gages to be edited/deleted, and definition of rainfall intensity curves by the *XY-Series Editor* in the same way it is defined for the uniform rainfall option.

The text window at the top of the gages dialog is used to specify which gage is currently active for editing. Clicking on the gage name within this window activates the gage and places its name and position in the edit fields.

Either inverse distance weighted interpolation, or the Thiessen polygon method of interpolation can be used when computing the influence of the gages on individual grid cells.

19.8 Editing CASC2D Maps

A *CASC2D* map refers to a file of values for any of the attributes which must be defined on a cell by cell basis. This includes all of the overland flow and infiltration parameters. *WMS* uses the same spreadsheet editor dialog for all such maps. This section describes the general operations of the map editing dialog, while the different maps which can be defined are discussed in the overland flow and infiltration properties sections.

The *Map Edit* dialog is shown in Figure 19-4. The text window on the left side of the dialog is used to select which map is currently active for editing. Map choices are determined by the toggle box or radio group choices selected just below the text window.

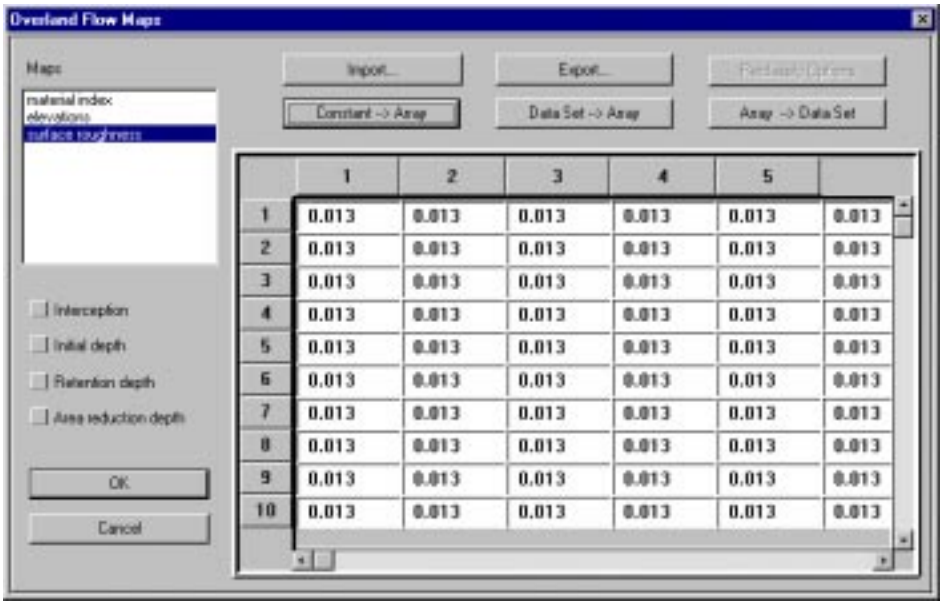


Figure 19-4 CASC2D Map Edit Dialog.

19.8.1 Importing a Map

The *Import button* can be used to import either an *ARC/INFO*® or GRASS ASCII grid file. The grid file should contain a single value for every cell. However, providing the imported grid covers the same aerial extent it can be of either higher or lower resolution.

19.8.2 Exporting a Map

The *Export* button allows the currently active map to be saved as either an *ARC/INFO*® or GRASS ASCII grid file. These files can be imported into either of these GIS's or read back into *WMS* at a later time.

19.8.3 Assigning a Constant to the Map

The *Constant->Array* button is used to set each value of the map to a constant value. After selecting this button the user is prompted for a constant value and then the entire map is updated with this value.

19.8.4 Creating a Data Set from a Map

The *Array->Dataset* button can be used to create a data set out of the currently active map. This is useful if you wish to contour any one of the parameters, or for performing mathematical computations on the entire array using the data calculator. For example *CASC2D* requires that all input parameters be entered in metric units, so the data calculator could be used to convert values of hydraulic conductivity from ft/sec to m/sec. Once the conversion operation is performed using the data calculator, the *Data set->Array* button can be used to convert the new data set values back to the map array. Values of hydraulic conductivity (or any other map parameter) could be contoured once the map is converted to a data set.

19.8.5 Creating a Map from a Data Set

The *Data set->Array* button can be used to convert a data set to a map for any of the available parameters. This is particularly useful if scattered data for a particular parameter is available and you wish to interpolate from the scattered data to a data set for the grid. The interpolated data set can then be used to create a map with this option. For example if hydraulic conductivity is known by filed observation at several points, then using a set of scattered points these values can be interpolated to the grid and then converted to a map for hydraulic conductivity. This feature is also useful for reassigning a data set to a map after the data calculator has been used to perform a mathematical operation as described in the previous section.

19.8.6 Reclassification

For most watersheds it will be difficult, if not impossible, to obtain cell by cell data for all of the parameters required to run a *CASC2D* model. While interpolation of scattered data is one solution for setting up maps, the reclassification method is often more practical and efficient. Reclassification allows a material or soil property id to be assigned to each cell. Then typical values of hydraulic conductivity, porosity or any/all of the required parameters can be defined for each different material. Finally reclassification causes all cells of a given material index to inherit that soils parameters for each map.

When the *Reclassify Options* button is chosen the dialog in Figure 19-5 comes up. This dialog is used to establish typical parameter values for each soil. A soil id and name must be defined for each soil. The toggle boxes at the top of

the dialog are undimmed if that map has been defined for the current model and dimmed if it is not. Turning on a toggle indicates that that value will be reclassified to the appropriate map when the Reclassify button is selected. For example if you wish to reclassify hydraulic conductivity, then one of the infiltration options must specified before entering the *Reclassify Options* dialog. The toggle for hydraulic conductivity must then be turned on and a value for each different material index defined. Finally selecting the reclassify button will cause the values of hydraulic conductivity to be mapped into the hydraulic conductivity according to the material index map.

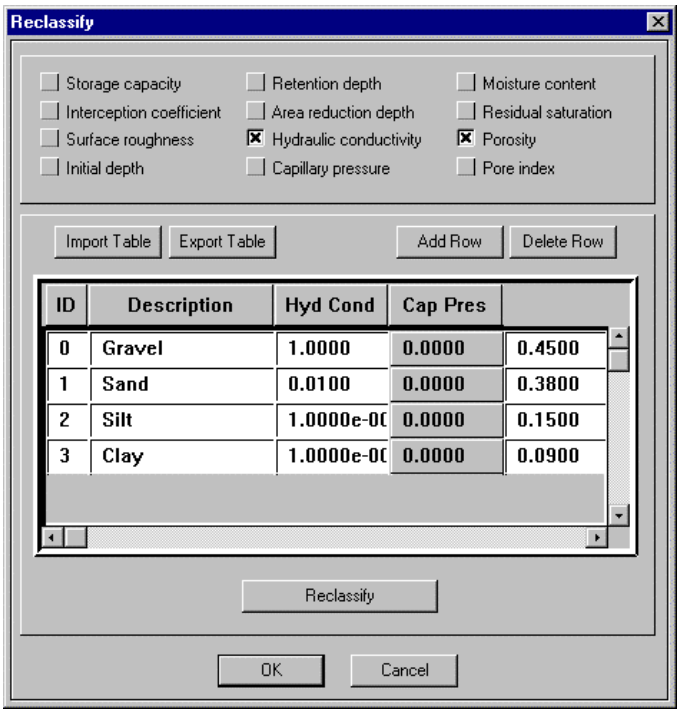


Figure 19-5 Reclassification Dialog.

A set of typical parameters for common soil types is distributed with *WMS* and can be used by importing the file soils.tbl. Once a set of typical parameters are defined they can be saved to a file for later recall using the export button. The format of the soil table file can be found in section 20.12.

19.9 Overland Flow Properties

The *CASC2D* maps are divided into three separate categories: overland flow, infiltration, and evapotranspiration. The maps related to overland flow are discussed in this section and those related to infiltration and evapotranspiration in the next two. Viewing/editing of maps is done using the *Edit Map* dialog as described in the previous section.

The overland flow options include elevation, surface roughness, interception coefficient and storage capacity, initial depth, retention depth, and area reduction depth. The elevation and surface roughness maps must be defined for all *CASC2D* models. All other maps represent optional parameters which can be used and are turned on or off for a given simulation using the toggle boxes which appear below the text window of the *Edit Map* dialog. Both the interception coefficient and storage capacity maps are activated when the interception option is toggled on.

- **Elevation** - This map is always required and specifies the elevation of each grid cell. This map is created automatically from a background elevation map when creating a new grid from a boundary polygon. For imported grids or grids created using the *Create Grid* command of the *Grid* menu, and elevation map must be imported.

- **Surface Roughness** - This map is required for all models. It represents the Manning's N roughness coefficient for each cell.

- **Storage Capacity** - This map is optional, however it must defined when the interception option is toggled on. It represents the vegetative storage capacity.

- **Interception Coefficient** - This map is optional and represents the vegetative interception coefficient.

- **Initial Depth** - This is a map of the initial overland flow depth and is optional. This map does not include lakes, the initial depth of lakes are specified separately.

- **Retention Depth** - This map is optional and specifies the retention depth.

- **Area Reduction Depth** - This map is optional and represents the area reduction for each grid cell.

19.10 Infiltration Properties

The infiltration options include no infiltration, Green-Ampt, and infiltration-redistribution. Specifying no infiltration means that none of the infiltration maps need be defined, however infiltration will not be modeled by *CASC2D*. Green-Ampt infiltration requires the hydraulic conductivity, capillary pressure head, porosity and moisture content maps to be defined. The infiltration redistribution option requires the same maps as Green-Ampt and the pore index and residual saturation maps as well. These two options treat infiltration the same, the difference is that infiltration-redistribution accounts for soil moisture recovery during periods of no rainfall.

- **Hydraulic Conductivity** - Map of hydraulic conductivity for each cell.
- **Capillary Pressure Head** -Map of capillary pressure heads.
- **Porosity** - Map of soil porosity for each cell.
- **Moisture** - Map of initial moisture content for each cell.
- **Pore Index** - Map of pore index values.
- **Residual Saturation** - Map of residual saturation.

With the exception of initial moisture content each of these maps is typically defined using the material index and reclassification as described in the *Edit Map* dialog section above. Initial moisture content should be defined using antecedent moisture considerations.

19.11 Evapotranspiration

The evapotranspiration options include no evapotranspiration calculations, Deardorff, and Penman. Specifying no evapotranspiration means that none of the evapotranspiration maps need be defined, however evapotranspiration will not be modeled by *CASC2D*. Deardorff evapotranspiration requires the land-surface albedo and wilting point water content maps to be defined. The Penman option requires these same two maps, plus vegetation height, vegetation radiation coefficient, and a canopy stomatal resistance maps to be defined as well.

- **Land-surface albedo**.
- **Wilting point water content**.
- **Vegetation height**.
- **Vegetation radiation coefficient**.
- **Canopy stomatal resistance**.

A more complete discussion of the evapotranspiration parameters is given in the *CASC2D* reference manual.

19.12 Editing Map Attributes

In addition to using the *Edit Map* dialog, all parameters for a set of selected cells can be edited using the *Cell Properties* command. Cells can be selected

using any one of the methods for cell selection, but at least one cell must be selected prior to specifying this option. If multiple cells are selected then the edit field for each parameter is grayed out until the first mouse click in that field. Only the fields which are not grayed are updated when the okay button is selected. If a field is accidentally activated and you wish to deactivate it once again hold down the shift key while clicking in the field.

19.13 Defining Channels

CASC2D can be run with or without hydraulic channel routing methods coupled with the overland flow computations. Channels are defined using feature arcs from the Map module. A complete discussion of these options is given Chapter 3. When writing *CASC2D* files the feature objects are rasterized onto the computational grid with the grid cells inheriting the attributes of the feature arc overlaying it. *CASC2D* uses a link and node map to identify which cells are in the channel.

19.14 The CASC2D Model Checker

The *Model Checker* command tries to identify any potential problems or data inconsistencies before the model project is saved and run through *CASC2D*. Checks are continually being added and the developers encourage suggestions from users who have struggled with a problem that could easily be identified by the model checker.

19.15 Running A CASC2D Model

Once all necessary data for a *CASC2D* simulation has been defined a project file can be saved and then used as input to the *CASC2D* program. *CASC2D* is distributed as a separate program from *WMS*, but can be launched from within *WMS*. The *Run CASC2D* command launches *CASC2D* and sends the given project file name as a command line argument. By default the project and all associated files are saved prior to executing *CASC2D* in order to insure that all changes made are saved to files.

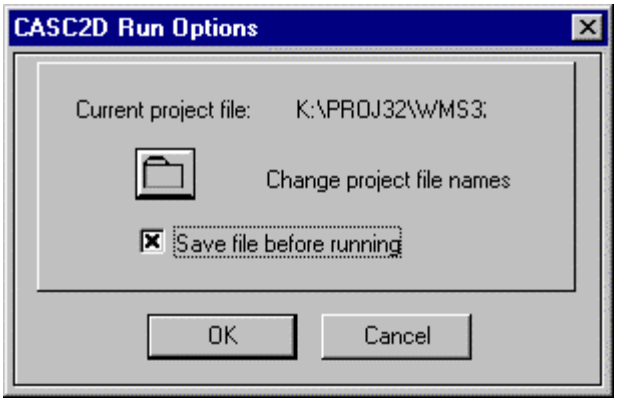


Figure 19-6 Run CASC2D Dialog.

As an alternative the project file can be saved using the *Save Project* command and then *CASC2D* run as a stand-alone process. When run as a stand-alone process the project file should be specified as a command line argument.

When using defined channels in combination with surface runoff, *CASC2D* must first be run to initialize depth and flows in the channel cells. This is done by specifying one of the channel initialization options in the *Job Control* and then running *CASC2D*. When doing initialization the discharge and water surface profile files are the only output. These files are subsequently used as input when a full channel routing simulation is performed.

19.16 Post-Processing Results

Post-processing of *CASC2D* models is done in *WMS* through the standard data commands. Any of the output maps available in *CASC2D* can be imported as data sets after a simulation has been successfully run. These output maps are toggled on from the *Output Control* dialog accessible from the *Job Control* command and file names are specified in the *Save Project* dialog. These maps are enumerated in the section describing the job control parameters.

Whenever a project file is imported, files specifying output maps are checked to see if they exist (i.e. the simulation has already been run and the maps generated). If they do they are automatically imported as data sets for the project.

Once imported, any of the post-processing options, including contours, animations, and gages can be used. These options are all described in Chapter 16 on data sets.

CHAPTER 20

File Formats

20.1 Introduction

This chapter contains the file formats for most of the files used by *WMS*. Files which are used by analysis codes such as *HEC-1* are not documented here since they are described in the documentation for the codes.

Most of the files used by *WMS* have a card type format. With this format, the different components of the file are grouped into logical groups called "cards." The first component of each card is a short name which serves as the identifier. The remaining fields on the line contain the information associated with the card. In some cases, such as lists, a card can use multiple lines.

There are many advantages associated with the card type approach to formatting files. Some of the advantages are:

1. Card identifiers make the file easier to read. Each input line has a label which helps to identify the data on the line.

2. The cards names are useful as text strings for searching in a large file. All input lines of a particular type can be located quickly in a large input file.

3. Cards allow the data to be input in any order in many cases, i.e., the order that the cards appear in the file is usually not important.

4. Cards make it easy to modify a file format. New data can be included simply by defining a new card type. If the new card is optional (which

is typically the case for new cards) old files are still compatible. If an old card type is no longer used, the card can simply be ignored without causing input errors.

A disadvantage of card style input is that the card identifiers make the file slightly more verbose.

20.2 *WMS* Super Files

A *WMS* super file is a file which contains a list of other files. If a super file is selected using the *Open* command in the *File* menu, each of the files listed in the super file are opened and imported. This makes it possible to quickly read in several files without having to identify each file individually in the file browser. Most of the files are files created by *WMS*, however other files such as DXF, and *HEC-1* files can be included in order to maintain all files pertaining to a given project together.

The super file format, with all of the possible options, is shown in

GRID2D FILE.2DG /* 2D Grid file */

```
SCATTER                 FILE.XY /* Scatter point file */
HEC1  FILE.HC1          /* HEC-1 file */
TR20  FILE.DAT          /* TR-20 file */
DATA  FILE.SCL          /* Data set file */
IMAGE FILE.IMG          /* Image registration file */
DXF   C:\WMS\DATA\FILE.DXF      /* DXF file */
```

Figure 20-1. The first line in the file is a SUPER card that identifies the file as a super file. Each of the other cards has a file type identifier followed by a file name. Except for DXF files, the file name should not contain the directory path. Any suffix may be used with the file name. A sample super file is shown in MAP

DXF C:\WMS\DATA\SITE.DXF

Figure 20-2.

```
SUPER                           /* Super file type identifier */
TIN   FILE.TIN          /* TIN file */
DEM   FILE.GRD          /* DEM file */
MAP   FILE.MAP          /* MAP file */
GRID2D                  FILE.2DG       /* 2D Grid file */
SCATTER                 FILE.XY /* Scatter point file */
HEC1  FILE.HC1          /* HEC-1 file */
TR20  FILE.DAT          /* TR-20 file */
DATA  FILE.SCL          /* Data set file */
IMAGE FILE.IMG          /* Image registration file */
DXF   C:\WMS\DATA\FILE.DXF      /* DXF file */
```

Figure 20-1. Super File Format.

```
SUPER
TIN   SITE.TIN
DEM   SITE.DEM
MAP   SITE.MAP
DXF   C:\WMS\DATA\SITE.DXF
```

Figure 20-2. Sample Super File.

20.3 TIN Files

TIN files are used for storing triangulated irregular networks. The TIN file format is shown in Figure 20-3 and a sample file is shown in Figure 20-4. The TIN file format can be used to import a simple set of XYZ coordinates since the triangle information (beginning with the TRI card) does not need to be present. If you have a file of xyz coordinates you only need to add the TIN, BEGT, and VERT nv cards to the top of the file and the ENDT card at the end.

```
TIN                     /* File type identifier */
BEGT                    /* Beginning of TIN group */
TNAM name               /* Name of TIN */
TCOL id                 /* TIN material id */
VERT nv                 /* Beg. of vertices */
x1 y1 z1 lf1            /* Vertex coords. */
x2 y2 z2 lf2
.
.
.
xnv ynv znv lfnv
TRI nt                  /* Beg. of triangles */
v11 v12 v13             /* Triangle vertices */
v21 v22 v23
.
.
.
vnt1 vnt2 vnt3
ENDT                    /* End of TIN group */
```

Figure 20-3. TIN File Format.

```
TIN
BEGT
TNAM Aspen
TCOL 255 255 255
VERT 408
0.0 3.1 7.8 0
5.3 8.7 4.0 1
.
.
2.4 4.4 9.0 1
TRI 408
5 1 4
4 1 2
.
.
4 2 3
ENDT
```

Figure 20-4. Sample TIN File.

The cards used in the TIN file are as follows:

Card Type	**TIN**
Description	File type identifier. Must be on first line of file. No fields.
Required	YES

Card Type	**BEGT**
Description	Marks the beginning of a group of cards describing a TIN. There should be a corresponding ENDT card at a latter point in the file. No fields.
Required	YES

Card Type	**TNAM**		
Description	Provides a name to be associated with the TIN.		
Required	NO		
Format	`TNAM name`		
Sample	`TNAM aspen`		
Field	*Variable*	*Value*	*Description*
1	name	str	The name of the TIN.

Card Type	**TCOL**		
Description	Defines a default color for the triangles of the TIN		
Required	NO		
Format	`TCOL color_red color_green color_blue`		
Sample	`TCOL 255 255 255`		
Field	*Variable*	*Value*	*Description*
1	color_red	0-255	The red color component of TIN triangles.
2	color_green	0-255	The green color component of TIN triangles.
3	color_blue	0-255	The blue color component of TIN triangles.

Card Type	**VERT**		
Description	Lists the vertices in the TIN		
Required	YES		
Format	`VERT nv` $x_1\ y_1\ z_1\ lf_1$ $x_2\ y_2\ z_2\ lf_2$ . . $x_{nv}\ y_{nv}\ z_{nv}\ lf_{nv}$		

Sample	`VERT 4`
	`0.0 3.1 7.8 0`
	`5.3 8.7 4.0 1`
	`2.4 4.4 9.0 1`
	`3.9 1.2 3.6 0`		

Field	*Variable*	*Value*	*Description*
1	nv	+	The number of vertices in the TIN
2-4	x,y,z	±	Coords. of vertex
5	lf	0,1	Locked / unlocked flag for vertex (optional). 0=unlocked, 1=locked. Repeat fields 2-5 nv times.

Card Type	**TRI**
Description	Lists the triangles in the TIN
Required	NO (a set of triangles can be generated from the vertices)

Format	`TRI nt`
	$v_{11}\ v_{12}\ v_{13}$
	$v_{21}\ v_{23}\ v_{23}$
	.
	.
	`vnt1 vnt2 vnt3`

Sample	TRI 4 5 1 4 4 1 2 4 2 3 5 4 3		
Field	*Variable*	*Value*	*Description*
1	nt	+	The number of triangles in the TIN.
2-4	v1,v2,v3	+	Vertices of triangle listed in a counter-clockwise order. Repeat nt times.

Card Type	**ENDT**
Description	Marks the end of a group of cards describing a TIN. There should be a corresponding BEGT card at a previous point in the file. No fields.
Required	YES

20.4 DEM Files

DEM files are used for storing DEMs processed by *WMS*. After clipping, thinning, or smoothing an imported DEM you may wish to save it to a *WMS* formatted file so that it can be recalled later without having to perform the same processing steps. The DEM file format is shown in Figure 20-5 and a sample file is shown in Figure 20-6.

```
DEM                        /* File type identifier */
ORIGIN xlowerleft ylowerleft    /* Southwest coordinare of DEM */
DELTAX deltx               /* X spacing of elevation points */
DELTAY delty               /* Y spacing of elevation points */
ELEVATIONS ncol nrow       /* Number of columns and rows in DEM */
z11                        /* elevation of row 1 column 1 */
z21                        /* elevation of row 2 column 1 */
z31                        /* elevation of row 3 column 1 */
.
.
.
zncol,nrow                 /* elevation of row nrow column ncol */
```

Figure 20-5. DEM File Format.

```
DEM
ORIGIN 1000.0 1500.0
DELTAX 30.0
DELTAY 30.0
ELEVATIONS 450 300
101
104
.
.
250
```

Figure 20-6. Sample DEM File.

The cards used in the DEM file are as follows:

Card Type	**DEM**
Description	File type identifier. Must be on first line of file. No fields.
Required	YES

Card Type	**ORDER**		
Description	Defines the order in which elevations are read in.		
Required	NO (By default row major starting in upper left corner is assumed)		
Format	`ORDER ordertype`		
Sample	`ORDER 0`		
Field	*Variable*	*Value*	*Description*
1	xlowleft	0-3	0 - Elevations start at upper left, one row at a time. 1 - Elevations start at upper left, one col. at a time. 2 - Elevations start at lower left, one row at a time. 3 - Elevations start at lower left, one col. at a time.

Card Type	**ORIGIN**		
Description	Defines the lower left (southwest) coordinates for the DEM.		
Required	YES		
Format	`ORIGIN xlowleft ylowleft`		
Sample	`TNAM 1000.0 1500.0`		
Field	*Variable*	*Value*	*Description*
1	xlowleft	±	Lower left X coordinate of the DEM
2	ylowleft	±	Lower left Y coordinate of the DEM

Card Type	**DELTAX**		
Description	The X spacing between DEM points.		
Required	YES		
Format	DELTAX deltx		
Sample	DELTAX 30.0		
Field	*Variable*	*Value*	*Description*
1	deltx	+	X spacing of DEM points.

Card Type	**DELTAY**		
Description	The Y spacing between DEM points.		
Required	YES		
Format	`DELTAY delty`		
Sample	`DELTAY 30.0`		
Field	*Variable*	*Value*	*Description*
1	delty	+	Y spacing of DEM points.

Card Type	**ELEVATIONS**		
Description	Elevations for the DEM.		
Required	YES		
Format	`ELEVATIONS ncol nrow`		

Sample	`ELEVATIONS 450 300`
	`101`
	`104`
	`.`
	`.`
	`.`
	`98`		

Field	*Variable*	*Value*	*Description*
1	ncol	+	Number of columns.
2	nrow	+	Number of rows.
3-n	z	+	Elevations of DEM points.

20.5 Image File

Image files are used in conjunction with TIFF files that have been previously imported to *WMS* and registered. They include the name of the TIFF file, the registration points, and the bounds of the clipping window. The format of the image file is shown in Figure 20-7 and a sample image file is shown in Figure 20-8.

```
IMAGE                    /* File type identifier */
TIFF "filename"          /* Indicates the name of the tiff file used */
IMREGPTS
PT1 u1 v1 x1 y1
PT2 u2 v2 x2 y2
PT3 u2 v2 x2 y2
CLIPPOINT
x1 x2
y1 y2
```

Figure 20-7 The Image File Format.

```
IMAGE
TIFF "jonescyn.tif"
IMREGPTS
PT1 0 756 422424.030700 4519460.893988
PT2 0 0 422424.030700 4515391.182075
PT3 715 0 426273.322285 4515391.182075
CLIPPOINT
422424.030700 426273.322285
4515391.182075 4519460.893988
```

Figure 20-8 Sample Image File.

The card types used in the Image file format are as follows:

Card Type	**IMAGE**
Description	File type identifier. Must be on first line of file. No fields.
Required	YES

Card Type	**TIFF**		
Description	Defines the name of the TIFF file to be displayed as an image.		
Required	YES		
Format	`TIFF "filename"`		
Sample	`TIFF "jonescyn.tif"`		
Field	*Variable*	*Value*	*Description*
1	filename	str	The name of the TIFF file.

Card Type	**PT1, PT2, PT3**		
Description	The three registration points used to define locations on a given image.		
Required	YES		
Format	PT1 tx_1 ty_1 wx_1 wy_1 PT2 tx_2 ty_2 wx_2 wy_2 PT3 tx_3 ty_3 wx_3 wy_3		
Sample	PT1 117 797 0.000000 10000.000000 PT2 117 88 0.000000 0.000000 PT3 1053 88 13220.0 0.000000		
Field	*Variable*	*Value*	*Description*
1-2 3-4	tx ty wx wy	± ±	Texture map coordinates. World coordinates.

Card Type	**CLIPPOINTS**		
Description	Defines the coordinates of the area in the TIFF file to be displayed as the image. (The area clipped and displayed from the TIFF file.		
Required	YES		
Format	CLIPPOINTS xmin xmax ymin ymax		
Sample	CLIPPOINTS -628.990382 14338.471657 -857.665608 8354.617436		
Field	*Variable*	*Value*	*Description*
1-2 3-4	xmin xmax ymin ymax	± ±	Min and max values in the x direction. Min and max values in the y direction.

20.6 2D Grid Files

Two-dimensional grids are stored in 2D grid files. The grids can be either cell-centered or mesh-centered. If the grid is mesh-centered, a set of material ids may be included in the file. The 2D grid file format is shown in Figure 20-9, and a sample file in Figure 20-10.

```
GRID2D          /* File type */
TYPE i          /* Type of grid. Mesh or Cell centered. */
IJ ±idir ±jdir  /* Card for defining rows, columns. */
DIM nx ny       /* # of cell boundaries in each direction. */
x1              /* X coord. of cell boundaries. */
x2
.
.
xnx
y1              /* Y coord. of cell boundaries. */
y2
.
.
yny
DELEV el        /* Default elevation for grid. */
```

Figure 20-9. 2D Grid File Format.

```
GRID2D
ID 5758
TYPE 1
DELEV 0.000000000000000e+00
IJ -y +x
DIM 4 4
0.000000000000000e+00
3.333333333333334e+01
6.666666666666667e+01
1.000000000000000e+02
0.000000000000000e+00
3.333333333333334e+01
6.666666666666667e+01
1.000000000000000e+02
```

Figure 20-10. Sample 2D Grid File.

The card types used in the 2D grid file format are as follows:

Card Type	**GRID2D**
Description	File type identifier. Must be on first line of file. No fields.
Required	YES

Card Type	**TYPE**		
Description	Defines the type of grid as either cell- or mesh-centered.		
Required	YES		
Format	`TYPE i`		
Sample	`TYPE 0`		
Field	*Variable*	*Value*	*Description*
1	i	0,1	The type code: i = 0 for mesh-centered i = 1 for cell-centered.

Card Type	**IJ**		
Description	Defines the orientation of the i,j indices.		
Required	YES		
Format	`IJ ±idir ±jdir`		
Sample	`IJ +x -y`		
Field	*Variable*	*Value*	*Description*
1	±idir	±x,±y	The direction corresponding to an increasing i index.
2	±jdir	±x,±y	The direction corresponding to an increasing j index.

Card Type	**DIM**		
Description	Defines the dimensions of the grid.		
Required	YES		
Format	`DIM nx ny` x_1 x_2 . . x_{nx} y_1 y_2 . . y_{ny}		

Sample	`DIM 4 6`
	`0.0`
	`1.0`
	`2.0`
	`4.0`
	`10.0`
	`12.0`
	`14.0`
	`16.0`
	`18.0`
	`20.0`		

Field	*Variable*	*Value*	*Description*
1	nx	+	The number of cell boundaries in the x direction.
2	ny	+	The number of cell boundaries in the y direction.
3 to (nx+2)	x_1-x_{nx}	±	The coordinates of the x boundaries.
(nx+3) to (nx+ny+2)	y_1-y_{ny}	±	The coordinates of the y boundaries.

Card Type	**DELEV**		
Description	Defines a default elevation for the grid.		
Required	NO		
Format	DELEV el		
Sample	DELEV 100.0		
Field	*Variable*	*Value*	*Description*
1	el	±	The default elevation.

20.7 2D Scatter Point Files

Two-dimensional scatter point sets are stored in 2D scatter point files. Multiple scatter point sets can be stored in a single file. Each point in a scatter point set is defined by a pair of xy coordinates. The format of the 2D scatter file is shown in Figure 20-11 and a sample file in Figure 20-12.

```
SCAT2D           /* File type identifier */
BEGSET           /* Beginning of cards for scatter point set */
NAME "name"      /* Name of scatter point set */
ID id            /* ID of scatter point set */
DELEV elev1      /* Default elevation */
IXY np           /* Number of points in set, begin point listing */
id1 x1 y1        /* Point id and coordinatess, one per line */
id2 x2 y2
.
.
idnp xnp ynp
ENDSET           /* End of cards for scatter point set */
/* Repeat point set cards as many times as necessary */
```

Figure 20-11 2D Scatter Point File Format.

```
SCAT2D
BEGSET
NAME   "gages"
ID   8493
DELEV 0.00000000000e+00
IXY 25
1 1.47000000000e+02 3.90000000000e+02
2 8.82000000000e+02 9.49000000000e+02
.
.
24 1.73000000000e+02 7.01000000000e+02
25 5.39000000000e+02 8.98000000000e+02
ENDSET
```

Figure 20-12 Sample 2D Scatter Point File.

The cards used in the 2D scatter point file are as follows:

Card Type	**SCAT2D**
Description	File type identifier. Must be on first line of file. No fields.
Required	YES

Card Type	**BEGSET**
Description	Identifies the beginning of a scatter point set. No fields.
Required	NO

Card Type	**NAME**		
Description	Defines the name for the following scatter point set.		
Required	NO		
Format	`NAME "name"`		
Sample	`NAME "wells"`		
Field	*Variable*	*Value*	*Description*
1	name	str	The name for the following scatter points. Remains as default until new NAME card is encountered.

Card Type	**ID**		
Description	Defines the ID for the scatter point set.		
Required	YES		
Format	`ID id`		
Sample	`ID 43098`		
Field	*Variable*	*Value*	*Description*
1	id	+	The ID for the following scatter point set.

Card Type	**DELEV**		
Description	Defines the default elevation for the scatter point set.		
Required	NO		
Format	`DELEV el`		
Sample	`DELEV 9.0`		
Field	*Variable*	*Value*	*Description*
1	el	±	The default elevation for the following scatter points. Remains as default until new DELEV card is encountered.

Card Type	**IXY**		
Description	Defines a scatter point set.		
Required	YES		
Format	`IXY np` id_1 x_1 y_1 id_2 x_2 y_2 . . id_{np} x_{np} y_{np}		

Sample	`IXY 4`
	`1 12.3 34.5`
	`2 52.2 23.5`
	`3 63.2 27.4`
	`4 91.1 29.3`		

Field	*Variable*	*Value*	*Description*
1	np	+	The number of scatter points in the scatter point set.
2	id	+	The ids of the points.
3-4	x,y	±	The coordinates of the points.
Repeat fields 2-4 np times			

Card Type	**ENDSET**
Description	Identifies the end of a scatter point set. No fields.
Required	NO

20.8 ASCII Data Set Files

Data sets can be stored to either ASCII or binary files. Multiple data sets can be stored in a single file and both scalar and vector data sets can be saved to the same file. The file format is identical for 2D and 3D data sets. The ASCII data set format is shown in Figure 20-13. A sample data set file is shown in Figure 20-14.

For scalar data set files, one value is listed per vertex, cell, node, or scatter point. For vector data set files, one set of XYZ vector components is listed per vertex, cell, node, or scatter point. If necessary, a set of status flags can be included in the file. If the status flag is false (0), the corresponding item (node, cell, etc.) is inactive. If status flags are not included in the file, it is assumed that all items are active.

```
DATASET          /* File type identifier */
OBJTYPE type     /* Type of object data set is associated with */
BEGSCL           /* Beginning of scalar data set */
OBJID id         /* Object id */
ND numdata       /* Number of data values */
NC numcells      /* Number of cells or elements */
NAME "name"      /* Data set name */
TS istat time    /* Time step of the following data. */
stat1            /* Status flags */
stat2
.
.
statnumcells
val1             /* Scalar data values */
val2
.
.
valnumdata
/* Repeat TS card for each time step */
ENDDS            /* End of data set */
BEGVEC           /* Beginning of vector dataset */
VECTYPE type     /* Vector at node/gridnode or element/cell */
OBJID id         /* Object id */
ND numdata       /* Number of data values */
NC numcells      /* Number of cells or elements */
NAME "name"      /* Data set name */
TS istat time    /* Time step of the following data. */
stat1            /* Status flags */
stat2
.
.
statnumcells
vx1 vy1 vz1
vx2 vy2 vz2
.
.
vnumdata vnumdata vnumdata
/* Repeat TS card for each time step */
ENDDS            /* End of data set */
/* Repeat BEGSCL and BEGVEC sequences for each data set */
```

Figure 20-13 ASCII Data Set File Format.

```
DATASET
OBJTYPE grid2d
BEGSCL
OBJID 27211
ND 8
NC 8
NAME "trichloroethylene"
TS 1 1.00000000e+00
0
0
0
1
1
1
1
0
0.00000000e+00
0.00000000e+00
0.00000000e+00
3.24000000e+00
4.39000000e+00
2.96000000e+00
7.48000000e+00
0.00000000e+00
ENDDS
BEGVEC
VECTYPE 0
OBJID 27211
ND 8
NC 8
NAME "velocity"
TS 1 5.00000000e+00
0
0
0
1
1
1
1
0
1.60000000e+01 1.60000000e+01 3.20000000e+01
6.40000000e+01 6.40000000e+01 1.28000000e+02
1.44000000e+02 1.44000000e+02 2.88000000e+02
1.96000000e+02 1.96000000e+02 3.92000000e+02
2.25000000e+02 2.25000000e+02 4.50000000e+02
9.21600000e+03 9.21600000e+03 1.84320000e+04
9.60400000e+03 9.60400000e+03 1.92080000e+04
9.80100000e+03 9.80100000e+03 1.96020000e+04
ENDDS
```

Figure 20-14 Sample ASCII Data Set File.

The card types used in the scalar data set file format are as follows:

Card Type	**DATASET**
Description	File type identifier. Must be on first line of file. No fields.
Required	YES

Card Type	**OBJTYPE**		
Description	Identifies the type of objects that the data sets in the file are associated with.		
Required	YES. If card does not exist, the file can only be read through the Data Browser. The data sets would then be assigned to the objects corresponding to the active module.		
Format	`OBJTYPE type`		
Sample	`OBJTYPE tin`		
Field	*Variable*	*Value*	*Description*

1	type	tin	TINs
		mesh2d	2D meshes
		grid2d	2D grids
		scat2d	2D scatter points
		mesh3d	3D meshes
		grid3d	3D grids
		scat3d	3D scatter points

Card Type	**BEGSCL**
Description	Scalar data set file identifier. Marks beginning of scalar data set. No fields.
Required	YES

Card Type	**BEGVEC**
Description	Vector data set file identifier. Marks beginning of vector data set. No fields.
Required	YES

Card Type	**VECTYPE**			
Card ID	150			
Description	Identifies the type of vector data that will be read and where to apply it.			
Required	This card is only required if the vector data is associated with elements/cells. If this card is not present, it is assumed that the data are associated with nodes/gridnodes.			
Field	*Variable*	*Size*	*Value*	*Description*
1	type	4 byte int	0	The vectors will be applied to the nodes/gridnodes.
			1	The vectors will be applied to the elements/cells.

Card Type	**ND**		
Description	The number of data values that will be listed per time step. This number should correspond to the total number of vertices, nodes, cells centers (cell-centered grid), cell corners (mesh-centered grid), maximum node id (meshes) or scatter points.		
Required	YES		
Format	`ND numdata`		
Sample	`ND 10098`		
Field	*Variable*	*Value*	*Description*
1	numdata	+	The number of items. At each time step, numdata values are printed.

Card Type	**NC**		
Description	This number should correspond to the maximum element id (meshes) or the number of cells (grids).		
Required	YES		
Format	`NC numcells`		
Sample	`NC 3982`		
Field	*Variable*	*Value*	*Description*
1	numcells	+	The number of elements or cells.

Card Type	**NAME**
Description	The name of the data set.

Required	YES		
Format	NAME "name"		
Sample	NAME "Total head"		
Field	*Variable*	*Value*	*Description*
1	"name"	str	The name of the dataset in double quotes.

Card Type	**TS**		
Description	Marks the beginning of a new time step, indicates if stat flags are given, and defines the time step value, status flags, and scalar data values for each item.		
Required	YES		
Format	TS istat time stat1 stat2 . . stat numcells val1 val2 . . valnumdata		
Sample	TS 1 12.5 0 1 1 1 34.5 74.3 58.4 72.9		
Field	*Variable*	*Value*	*Description*
1	istat	0 1	Use status flags from previous time step. For first time step, this indicates that all cells are active. Status flags will be listed.
2	time	+	The time step value. If only one time step exists, time is not required
2 - (n+1)	stat	0,1	The status of each item. If active, stat=1. If inactive stat=0. Omitted if i=0 on STAT card.
(n+2) - (2n+1)	val	±	The scalar data values of each item.

20.8.1 Using Vector and Scalar Data with Grids

Vector and scalar data can be associated with the nodes or cells of a 2D grid. The order that the scalar or vector values appear in the data set file determine which nodes or cells the data are associated with. For 2D grids, data values are ordered using a row-column (I-J) priority.

The following C source code examples illustrate how a 2Darray of scalar values corresponding to the nodes of a grid would be written to the main portion of an ASCII scalar file.

2D Grid Example:

```
for(i=0; i<nrow; i++){
  for(j=0; j<ncol; j++){
    fprintf(fp, "%f\n", scalar[i][j]);
  }
```

```
}
```

20.9 Binary Data Set Files

Data sets can be stored to either ASCII or binary files. Compared to ASCII files, binary files require less memory and can be imported to *WMS* more quickly. The disadvantages of binary files are that they are not as portable and they cannot be viewed with a text editor.

The binary data set file format is shown in Figure 20-15. The binary format is patterned after the ASCII format in that the data are grouped into "cards". However, the cards are identified by a number rather than a card title.

Card	Item	Size	Description
	version	4 byte integer	The GMS binary data set file format version. value = 3000.
100	objecttype	4 byte integer	Identifies the type of objects that the data sets in the file are associated with. Options are as follows: 1 TINs 2 Boreholes 3 2D meshes 4 2D grids 5 2D scatter points 6 3D meshes 7 3D grids 8 3D scatter points
110	SFLT	4 byte integer	The number of bytes that will be used in the remainder of the file for each floating point value (4, 8, or 16).
120	SFLG	4 byte integer	The number of bytes that will be used in the remainder of the file for status flags.
130 or 140	BEGSCL or BEGVEC		Marks the beginning of a set of cards defining a scalar or vector data set.
150	VECTYPE	4 byte integer	(0 or 1) In the case of vector data set files, indicates whether the vectors will be applied at the nodes/gridnodes or the elements/cells.
160	OBJID	4 byte integer	The id of the associated object. Value is ignored for grids and meshes.
170	NUMDATA	4 byte integer	The number of data values that will be listed per time step. This number should correspond to the number of vertices, nodes, cell centers (cell-centered grid), cell corners (mesh-centered grid) or scatter points.
180	NUMCELLS	4 byte integer	This number should correspond to the number of elements (meshes) or the number of cells (mesh-centered grids). Value is ignored for other object types.
190	NAME	40 bytes	The name of the dataset. Use one character per byte. Mark the end of the string with the '\0' character.
200	TS		Marks the beginning of a time step.
	ISTAT	SFLG integer	(0 or 1) Indicates whether or not status flags will be included in the file.
	TIME	SFLT real	Time corresponding to the time step.

	statflag1	SFLG integer	Status flag (0 or 1) for node 1
	statflag2	SFLG integer	Status flag (0 or 1) for node 2
	….		
	val1	SFLT real	Scalar value for item 1
	val2	SFLT real	Scalar value for item 2
	….		
			Repeat card 200 for each timestep in the data set.
210	ENDDS		Signal the end of a set of cards defining a data set.

Figure 20-15 The Binary Scalar or Vector Data Set File Format.

The cards in the binary data set file are as follows:

Card Type	**VERSION**
Card ID	3000
Description	File type identifier. No fields.
Required	YES

Card Type	**OBJTYPE**			
Card ID	100			
Description	Identifies the type of objects that the data sets in the file are associated with.			
Required	YES. If card does not exist, the file can only be read through the Data Browser. The data sets would then be assigned to the objects corresponding to the active module.			
Field	*Variable*	*Size*	*Value*	*Description*
1	id	4 byte int	1	TINs
			2	Boreholes
			3	2D meshes
			4	2D grids
			5	2D scatter points
			6	3D meshes
			7	3D grids
			8	3D scatter points

Card Type	**SFLT**			
Card ID	110			
Description	Identifies the number of bytes that will be used in the remainder of the file for each floating point value (4, 8, or 16).			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	sizefloat	4 byte int	4, 8, or 16	Number of bytes

Card Type	**SFLG**			
Card ID	120			
Description	Identifies the number of bytes that will be used in the remainder of the file for status flags (1, 2, or 4).			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	sizeflag	4 byte int	1, 2, or 4	Number of bytes

Card Type	**BEGSCL**

Card ID	130
Description	Marks the beginning of a set of cards defining a scalar data set.
Required	YES

Card Type	**BEGVEC**
Card ID	140
Description	Marks the beginning of a set of cards defining a vector data set.
Required	YES

Card Type	**VECTYPE**			
Card ID	150			
Description	Identifies the type of vector data that will be read and where to apply it.			
Required	This card is only required if the vector data is associated with elements/cells. If this card is not present, it is assumed that the data are associated with nodes/gridnodes.			
Field	*Variable*	*Size*	*Value*	*Description*
1	type	4 byte int	0 1	The vectors will be applied to the nodes/gridnodes. The vectors will be applied to the elements/cells.

Card Type	**OBJID**			
Card ID	160			
Description	The id of the associated object.			
Required	This card is required in the case of TINs, 2D scatter points, and 3D scatter points. With each of these objects, multiple objects may be defined at once. Hence the id is necessary to relate the data set to the proper object.			
Field	*Variable*	*Size*	*Value*	*Description*
1	id	4 byte int	+	The id of the object.

Card Type	**NUMDATA**			
Card ID	170			
Description	The number of data values that will be listed per time step. This number should correspond to the number of vertices, nodes, cell centers (cell-centered grid), cell corners (mesh-centered grid), maximum node id (meshes) or scatter points.			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	numdata	4 byte int	+	The number of items. At each timestep, numdata are listed.

Card Type	**NUMCELLS**			
Card ID	180			
Description	This number should correspond to the element id (meshes) or the number of cells (grids).			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	numcells	4 byte int	+	The number of elements or cells.

Card Type	**NAME**
Card ID	190

Description	The name of the data set.			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	name	40 bytes	str	The name of the data set. Use one character per byte. Mark the end of the string with the '\0' character.

Card Type	**TS**			
Card ID	200			
Description	Defines the set of scalar values associated with a timestep. Should be repeated for each time step.			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	istat	SFLG int	0 1	Use status flags from previous time step. For the first time step, this value indicates that all cells are active. Status flags will be listed.
2	time	SFLT int	+	The time step value. This number is ignored if there is only one time step.
	stat	SFLG int	0 1	Inactive Active One status flag should be listed for each cell or element. These flags are included only when istat = 1.
	val	SFLT real	±	The scalar values

Card Type	**ENDDS**
Card ID	210
Description	Signals the end of a set of cards defining a data set
Required	YES

20.10 GRASS Grid Files (CASC2D Maps)

WMS can import GRASS ASCII grid for use as a background DEM. Since it is a simple file format other digital elevation data can be formatted in the same way and then imported into *WMS* using the *Import Grid* command in the Dems menu.

The *CASC2D* model also uses GRASS ASCII grid files format for all of the map parameters. The GRASS ASCII format is shown in Figure 20-16, and an example of file is shown in Figure 20-17.

```
north:             n     /* Northern boundary grid coordinate */
south:             s     /* Southern boundary grid coordinate */
east: e            /* Eastern boundary grid coordinate */
west: w            /* Western boundary grid coordinate */
rows: nrows        /* Number of rows in the grid */
cols: ncols        /* Number of columns in the grid */
z11 z12 z13 ... z1ncols /* values of row 1 */
z21 z22 z23 ... z2ncols /* values of row 2 */
.
.
.
znrows1 znrows2 znrows3 ... znrowsncols /* values of last row*/
```

Figure 20-16. GRASS ASCII Grid File.

```
north: 3451250
south: 3438850
east: 298960
west: 290860
rows: 10
cols: 5
0 1 1 0 1
1 1 0 0 1
.
.
0 0 1 1 0
```

Figure 20-17. Sample GRASS ASCII Grid File.

The card types used in the GRASS grid file format are self explanatory.

20.11 *ARC/INFO*® ASCII Grid Files

WMS can import *ARC/INFO*® ASCII grid for use as a background DEM. Since it is a simple file format other digital elevation data can be formatted in the same way and then imported into *WMS* using the *Import Grid* command in the Dems menu.

The *CASC2D* model may also import and used *ARC/INFO*® grid files when defining map parameters. The file format is shown in Figure 20-18 and an example file in Figure 20-19.

```
ncols         ncol       /* Number of columns in the grid */
nrows         nrow       /* Number of rows in the grid */
xllcorner     x          /* Lower left x coordinate of grid */
yllcorner     y          /* Lower left x coordinate of grid */
cellsize      size       /* Grid cell size */
NODATRA_valueNODATA      /* value of an empty grid cell */
z11 z12 z13 ... z1ncols /* values of row 1 */
```

```
z21 z22 z23 ... z2ncols /* values of row 2 */
.
.
.
znrows1 znrows2 znrows3 ... znrowsncols /* values of last row*/
```

Figure 20-18. ARC/INFO® *ASCII Grid File Format.*

```
ncols         128
nrows         136
xllcorner     422415
yllcorner     4515405
cellsize      30
NODATA_value  -9999
1287 1286 1286 1288 ...
1288 1288 -9999 1289 ...
.
.
1282 -9999 1283 1284 ...
```

Figure 20-19. Sample ARC/INFO® *ASCII Grid File.*

The card types used in the *ARC/INFO*® grid file format are self explanatory.

20.12 CASC2D Soils Table File

A soils table file is used to reclassify CASC2D grid cell soil parameters from a look up table comparing soil types with typical values for the desired soil properties. The are created in *WMS* from the Reclassification dialog accessible when editing CASC2D maps. The first portion of the file contains the property types defined in the table.

Following the initial definitions there is one record for each soil in the table. The number and order of the values following the id and description of each soil is determined by the number and order of the property definitions defined in the first section. The example shown in Figure 20-20 demonstrates a file where all property values are present.

```
SOILSTABLE              /* Soils table file identifier */
HYDRAULIC_CONDUCTIVITY  /* Hydraulic conductivity in first field */
CAPILLARY               /* Capillary pressure head in second field */
POROSITY                /* Porosity in third field */
PORE_INDEX              /* Pore index in fourth field */
RESIDUAL_SATURATION     /* Residual saturation in fifth field */
MOSITURE_CONTENT             /* Moisture content in sixth field */
SURFACE_ROUGHNESS            /* Manning's n in seventh field */
INTERCEPTION_COEFFICIENT/* Interception coefficient in eighth field */
STORAGE_CAPACITY             /* Storage capacity in ninth field */
INITIAL_DEPTH           /* Initial depth values in tenth field */
RETENTION               /* Storage retention values in eleventh field */
AREA_REDUCTION               /* Area reduction factor in twelfth field
*/
id1 "description1" hc1 ch1 psty1 pi1 rs1 mc1 sr1 ic1 sc1 id1 ret1 ar1
id2 "description2" hc2 ch2 psty2 pi2 rs2 mc2 sr2 ic2 sc2 id2 ret2 ar2
.
.
.
id„ "description„" hc„ ch„ psty„ pi„ rs„ mc„ sr„ ic„ sc„ id„ ret„ ar„
```

Figure 20-20. CASC2D Soils Table File Format.

The order of the definition cards determines the order of the property values. Any property can be omitted from the definitions and it's value dropped for each soil record. In other words the soils table file is self-describing.

The card types used in the Soils table file format are self explanatory and are not listed here.

20.13 Gage Files

Gages are used to generate curves representing the variation of a transient data set with time at particular location in a grid. Gage plots are useful in the process of calibrating a model to measured field data. Gage files are used to import a set of gages and a set of measured curves at each gage. This allows a set of measured curves to be compared to a set of computed curves.

The format of the gage file is shown in Figure 20-21, and a sample is given in Figure 20-22. Several gages can be included in a single file. In addition, several vector or scalar curves can be associated with each gage. The times for each of the curves do not need to match.

```
GAGE              /* File type identifier */
BEGGAG            /* The beginning of a gage group. */
NAM name          /* The name of the gage. */
LOC x y z         /* The gage location. */
DIR vx vy vz      /* The positive direction vector. */
COL red green blue /* The color of the gage. */
SCA np name       /* A scalar curve. */
t1 v1             /* Time and value, from 1 to np. */
t2 v2
.
.
tnp vnp
VEC np name       /* A vector curve. */
t1 vx1 vy1 vz1    /* Time vector, from 1 to np. */
t2 vx2 vy2 vz2
.
.
tnp vxnp vynp vznp
/* Repeat scalar and vector cards as many times as necessary. */
ENDGAG            /* The end of a gage group */
/* Repeat gage card group as many times as necessary. */
```

Figure 20-21. Gage File Format.

```
GAGE
BEGGAG
NAM "Gage"
LOC 10232.768000 3097.442316.600 600.000000
DIR 0.000000 1.000000 0.000000
COL 0 255 51
SCA 10 Runoff
3.000000000000000e+02 5.960180683371563e-05
6.000000000000000e+02 9.733039686961903e-03
9.000000000000000e+02 5.495696626058331e-02
1.200000000000000e+03 6.660598699031131e+00
1.500000000000000e+03 2.132752979529125e+01
1.800000000000000e+03 3.714465098671455e+01
2.100000000000000e+03 5.304609314079730e+01
2.400000000000000e+03 6.622922014423446e+01
2.700000000000000e+03 6.744370275158799e+01
```

```
3.000000000000000e+03 6.702960411182080e+01
ENDGAG
```

Figure 20-22 Sample Gage File

The card types used in the gage file format are as follows:

Card Type	**GAGE**
Description	File type identifier. Must be on first line of file. No fields.
Required	YES

Card Type	**BEGGAG**
Description	Marks the beginning of a group of cards describing a gage. There should be a corresponding ENDGAG card at a latter point in the file. No fields.
Required	YES

Card Type	**NAM**		
Description	Defines a name to be associated with the gage.		
Required	NO		
Format	NAM name		
Sample	NAM Gage #23GP		
Field	*Variable*	*Value*	*Description*
1	name	str	The name of the gage

Card Type	**LOC**		
Description	Defines the xyz coordinates of the gage.		
Required	YES		
Format	LOC x y z		
Sample	LOC 348.23 623.36 93.2		
Field	*Variable*	*Value*	*Description*
1-3	x, y, z	±	The coordinates of the gage.

Card Type	**DIR**		
Description	Defines the positive direction vector for the gage. This vector is used to determine the sign (+ or -) when plotting vector curves.		
Required	NO		
Format	DIR vx vy vz		
Sample	DIR 0.45 0.55 0.0		
Field	*Variable*	*Value*	*Description*
1-3	vx, vy, vz	±	The components of the positive direction vector. The vector does not need to be normalized.

Card Type	**COL**		
Description	Identifies a color to be associated with the gage.		
Required	NO		
Format	COL id red green blue		
Sample	COL 124 67 245		
Field	*Variable*	*Value*	*Description*
1	red	0-255	The value of the red component of the color.
2	green	0-255	The value of the green component of the color.
3	blue	0-255	The value of the blue component of the color.

Card Type	**SCA**		
Description	Defines a curve of scalar values		
Required	NO		
Format	SCA np name t_1 v_1 t_2 v_2 . . t_{np} v_{np}		
Sample	SCA 6 head 0.0 100.0 0.5 99.93 1.0 99.14 1.5 98.64 2.0 98.32 2.5 98.12		
Field	*Variable*	*Value*	*Description*
1	np	+	The number of points in the curve.
2	name	str	The name of the curve. If curves of the same type
3	t	±	The time value for a point on the curve.
4	v	±	The scalar value for a point on the curve.
Fields 3-4 should be repeated np times.			

Card Type	**ENDGAG**
Description	Marks the end of a group of cards describing a gage. There should be a corresponding BEGGAG card at a previous point in the file. No fields.
Required	YES

20.14 Digitized Stream Files

Digitized streams can be used to create feature objects in *WMS*. A stream file contains several x-y coordinates for stream nodes. Z values can be present but are not required. A stream file could be created using any digitizing program, but the format of the file must be converted to the format outlined in Figure 20-23, and an actual sample in Figure 20-24.

```
STREAM                  /* Beginning of polygon stream file */
x1 y1 (z1)              /* Z values are optional */
x2 y2 (z2)
.
.
xnp ynp (znp)
```

Figure 20-23. Stream File Format.

```
STREAM
0.0 0.0
10.0 5.0
8.0 10.0
15.0 7.0
20.0 12.0
```

Figure 20-24. Sample Stream File.

The format of the STREAM card is as follows:

Card Type	**STREAM**		
Description	Defines a set of digitized points to be converted to feature points.		
Required	YES		
Format	STREAM x_1 y_1 (z_1) x_2 y_2 (z_2) . . x_n y_n (z_n)		
Sample	STREAM 0.0 0.0 10.0 5.5 8.0 10.0 15.0 7.0		
Field	*Variable*	*Value*	*Description*
1-2(3)	x,y,(z)	±	Stream coordinates. Repeat for each stream point. Only feature points are created from a stream file so the order of stream points in the file is not important.

20.15 AutoCAD Stream File Format

Some vendors of AutoCAD provide a utility for saving breakline files. This file format has been adopted in *WMS* so that arcs can be created from feature points as they are read in. The format is essentially the same as the digitized stream file except that there is no header card and there is an extra space in front of all points. This space is empty except for those points where a new stream segment begins in which case an S should be present. When reading these files a snapping tolerance can be defined so that all points preceded with the S can be "snapped" into the network. Remember that points should be defined from down to upstream. The file format is show in Figure 20-25, and a sample file in Figure 20-26.

```
STREAM                  /* Beginning of polygon stream file */
Sx11 y11 (z11)          /* First point in first stream segment */
 x112 y112 (z112)
.
.
 x1np y1np (z1np)
S x21 y21 (z21)         /* First point in first stream segment */
  x22 y22 (z22)
.
.
 x2np y2np (z2np)
```

Figure 20-25. AutoCAD Stream File Format.

```
S0.0 0.0
 10.0 5.5
 8.0 10.0
```

```
 15.0 7.0
```

Figure 20-26. Sample AutoCAD Stream File.

Format	Sx1 y1 (z1) x2 y2 (z2) . . xn yn (zn)		
Sample	S0.0 0.0 10.0 5.5 8.0 10.0 15.0 7.0		
Field	*Variable*	*Value*	*Description*
1-3(4)	x,y,(z)	±	Stream coordinates. Repeat for each stream point. Points preceded by an S are starting points of a new branch

20.16 Stage Files

Stage files are used to define water surface or stage values at locations on the TIN. These values are then used to automatically delineate flood plains. Values can be given in either absolute water surface elevation (z elevation plus water depth) or as stage (height of water above stream bed). The format of the stream file is shown in Figure 20-27, and a sample file in Figure 20-28.

```
FLOOD                    /* file identification string */
type                     /* value identification type */
x1, y1, f1               /* x, y position and stage or
x2, y2, f2                  water surface elevation value */
.
.
.
xn, yn, fn               /* Nth value */
```

Figure 20-27. Stream Stage File Format.

```
flood
stage
ST 153.749310 349.058500 10.000000
ST 329.571990 386.106990 10.000000
ST 614.905030 307.131010 10.000000
ST 822.443970 306.372990 10.000000
ST 441.833560 360.680420 10.000000
ST 495.800990 312.377010 10.000000
ST 763.363530 322.537260 10.000000
ST 678.969730 344.108950 10.000000
```

Figure 20-28. Sample Stream Stage File.

The format of the FLOOD card is as follows:

Card Type	**FLOOD**
Description	Defines a set of flood stage or water surface elevation values.
Required	YES
Format	FLOOD STAGE

	x_1 y_1 f_1 x_2 y_2 f_2 . . x_n y_n f_n		
Sample	FLOOD STAGE 100.0 250.0 5.5 300.0 450.0 8.0 275.0 800.0 10.2 330.0 740.0 15.0		
Field	*Variable*	*Value*	*Description*
1	type	str	The type of flood values used for f. Either water surface elevation (WATER) or stage (STAGE) values can be defined.
2-3	x, y	±	X, Y position of the stage value.
4	f	+	Flood stage or water surface elevation. Fields 2-4 are repeated for each flood value.

It should be noted that X and Y coordinates need not be exactly the same as they are represented in the TIN. The program automatically selects the closest vertex to the values found in the file and assigns the corresponding stage value to it. However, if a vertex has been moved and another vertex is now closer to the XY coordinates found in the file, the closer vertex will be assigned the stage value. Some manual editing of stage values may be needed to ensure proper results.

20.17 Land Use Files

When using a land use coverage or grid to map model parameters such as curve number, percent impervious, etc. a corresponding mapping file must be either created manually or imported into *WMS*. *WMS* will also export a mapping file created manually so that the same definitions can be remapped in a future model.

Depending on the application, one of three different land use tables will be required:

1. Mapping of land use to CN for hydrologic soil types A, B, C, and D.

2. Mapping of Green & Ampt parameters for the Maricopa County method.

3. Mapping of CASC2D attributes.

Descriptions and examples of each are given below.

Land use tables with corresponding curve numbers for different hydrologic soil groups vary from one text to another, or from one agency to another. For this reason, *WMS* supports user-defined tables. Users can create tables with the currently required data. The format of the table is shown in Figure 20-29, and a sample file in Figure 20-30.

```
ID1, "Land use description 1", CNA1, CNB1, CNC1, CND1
```

```
ID2, "Land use description 2", CNA2, CNB2, CNC2, CND2
ID3, "Land use description 3", CNA3, CNB3, CNC3, CND3
.
.
.
IDn, "Land use description n", CNAn, CNBn, CNCn, CNDn
```

Figure 20-29 Curver Number Land Use File Format.

```
1, "Fully developed urban areas, Poor Condtion", 68, 79, 86, 89
2, "Paved parking lots, roofs, driveways, etc.", 98, 98, 98, 98
3, "Residential 1/8 acre lots (65% impervious)", 77, 85, 90, 92
4, "Residential 1/3 acre lots (30% impervious)", 57, 72, 81, 86
```

Figure 20-30 Sample Curve Number Land Use File.

No specific card types are required in this file but a description of each of the six fields required for each land use definition is given below. Each field must be separated by a comma, and the description string must be enclosed by double quotes.

Field	*Variable*	*Value*	*Description*
1	id	+	ID number of land use description.
2	description	str	Land use description.
3-6	curve#	+	SCS Curve Number (CN) for hydrologic soil groups A,B,C,D.

WMS can be used to map Green & Ampt infiltration parameters for *HEC-1* using the Maricopa County methods. The file format and an example are given in Figure 20-31 and Figure 20-32.

```
ID1, "Land use description 1", IAB1, RTIMP1, PCTVEG1
ID2, "Land use description 2", IAB2, RTIMP2, PCTVEG2
ID3, "Land use description 3", IAB3, RTIMP3, PCTVEG3
.
.
.
IDn, "Land use description n", IABn, RTIMPn, PCTVEGn
```

Figure 20-31 Green & Ampt Land Use File.

```
20, "Mountainous shurb and brush", 0.3, 15.0, 50.0
19, "Mountainous forest", 0.25, 30.0, 50.0
29, "Mountainous grassland", 0.15, 55.0, 60.0
7, "Roadway", 0.3, 15.0, 50.0
```

Figure 20-32 Sample Green & Ampt Land Use File.

CASC2D map parameters may also be mapped to grid cells using a coverage or grid with an accompanying mapping file. The format of this file is given in Figure 20-33. The first part of the file serves as a dictionary. You may select any number of the "mappable" attributes listed, but values in the lower part of the table must appear in the same order as they are listed in the dictionary. An example table file is shown in Figure 20-34.

```
SOILSTABLE             /* File identifier */
HYDRAULIC_CONDUCTIVITY /* First field identifier */
CAPILLARY              /* Second field identifier */
POROSITY               /* Third field identifier */
PORE_INDEX             /* Fourth field identifier */
RESIDUAL_SATURATION    /* Fifth field identifier */
MOISTURE_CONTENT       /* Sixth field identifier */
SURFACE_ROUGHNESS      /* Seventh field identifier */
INTERCEPTION_COEFF     /* Eighth field identifier */
STORAGE_CAPACITY       /* Ninth field identifier */
```

```
INITIAL_DEPTH          /* Tenth field identifier */
RETENTION              /* Eleventh field identifier */
AREA_REDUCTION         /* Twelfth field identifier */
ALBEDO                 /* Thirteenth field identifier */
WILTING_POINT          /* Fourteenth field identifier */
VHEIGHT                /* Fifteenth field identifier */
TCOEFF                 /* Sixteenth field identifier */
CANOPY                 /* Seventeenth field identifier */
SOIL_ERODABILITY       /* Eighteenth field identifier */
CROP_MANAGEMENT        /* Nineteenth field identifier */
CONSERVATION_PRACTICE  /* Twentieth field identifier */
SAND_MAP               /* Twenty-first field identifier */
SILT_MAP               /* Twenty-second field identifier */
NUMSOILS n             /* Number of soils types in this file */
id1 "description1" hc1 cp1 po1 pi1 rs1 mc1 sr1 ic1 sc1 id1 rt1 ar1 ab1
wp1 vh1 tc1 ca1 se1 cm1 co1 sm1 si1
id2 "description2" hc2 cp2 po2 pi2 rs2 mc2 sr2 ic2 sc2 id2 rt2 ar2 ab2
wp2 vh2 tc2 ca2 se2 cm2 co2 sm2 si2
.
.
.
idn "descriptionn" hcn cpn pon pin rsn mcn srn icn scn idn rtn arn abn
wpn vhn tcn can sen cmn con smn sin
```

Figure 20-33 CASC2D Attribute Mapping File Format.

```
SOILSTABLE
HYDRAULIC_CONDUCTIVITY
CAPILLARY
POROSITY
PORE_INDEX
RESIDUAL_SATURATION
MOISTURE_CONTENT
SURFACE_ROUGHNESS
NUMSOILS 4
1 "Loamy" 0.001000 0.010000 0.100000 0.250000 0.100000 1.000000 0.200000
2 "Sandy" 0.002000 0.010000 0.100000 0.250000 0.100000 1.000000 0.100000
3 "Silty" 0.003000 0.010000 0.100000 0.250000 0.100000 1.000000 0.050000
4 "Clay" 0.004000 0.010000 0.100000 0.250000 0.100000 1.000000 0.030000
```

Figure 20-34 Sample CASC2D Attribute Mapping File.

20.18 Soil Type Runoff Coefficient Files

Like land use, soil type coverages/grids can be used to map model parameters. Tables with corresponding runoff coefficients, Green & Ampt parameters, or CASC2D values for different can be mapped. The CASC2D tables are the same as defined in the land use section above. Runoff coefficient table definitions and examples are given in Figure 20-35 and Figure 20-36.

```
ID1, "Soil type description 1", RC1
ID2, "Soil type description 2", RC2
ID3, "Soil type description 3", RC3
.
.
.
IDn, "Soil type description n", RCn
```

Figure 20-35. Soil Type Runoff Coefficient File Format.

```
1, "Highly impervious", 1.0
2, "Slightly pervious", .9
3, "Moderately pervious", .75
4, "Highly pervious", .5
```

Figure 20-36. Sample Soil Type Runoff Coefficient File.

Green & Ampt mapping file definitions and an example are given in Figure 20-37 and Figure 20-38.

```
ID1, "Soil type description 1", XKSAT1, RTIMP1, PCTEFFECTIVE1
ID2, "Soil type description 2", XKSAT2, RTIMP2, PCTEFFECTIVE2
ID3, "Soil type description 3", XKSAT3, RTIMP3, PCTEFFECTIVE3
.
.
.
IDn, "Soil type description n", XKSATn, RTIMPn, PCTEFFECTIVEn
```

Figure 20-37 Green & Ampt Soil Table File Format.

```
19, "Gunsight-Cipriano complex, 1-7% Slope", 0.63, 50.0, 0.0
18, "Greyeagle-Suncity Variant complex, 1-7% Slope", 0.23, 50.0, 0.0
17, "Gachado-Lomitas-Rock outcrop complex, 7-55% Slope", 0.16, 50.0, 20.0
72, "Lehmans-Rock outcrop complex, 8-65% Slope", 0.09, 50.0, 30.0
```

Figure 20-38 Sample Green & Ampt Soil Table File.

20.19 Hydrograph Files

When running *HEC-1*, *WMS* reads the TAPE22 hydrograph results file. However, it is convenient to read measured hydrographs for comparison with computed hydrographs, therefore a simplified hydrograph file format can be used. The format is shown in Figure 20-39, with a sample file given in Figure 20-40.

```
HYDROGRAPH
name   starttime   interval   numordinates
flow-1  flow-2  flow-3  ...  flow-10
...
...
...
flow-n
```

Figure 20-39. Hydrograph File Format.

```
HYDROGRAPH
B1   0   30   25
0.0  0.4  0.9  1.5  2.2  3.0  3.9  4.9  6.0  7.2
9.5  12.8  16.0  15.2  14.1  10.2  9.0  7.5  5.8  4.0
3.3  2.2  1.0  0.5  0.1
```

Figure 20-40. Sample Hydrograph File.

Card Type	**HYDROGRAPH**
Description	Defines a set of hydrographs.
Required	YES
Format	HYDROGRAPH name starttime interval numordinates flow-1 flow-2 flow-3 ... flow-10 ... ... flow-n
Sample	HYDROGRAPH B1 730 45 12 8.0 14.0 25.0 40.0 65.0 63.0 55.0 50.0 42.0 30.0 15.0 0.0

	B2 730 45 12 10.0 17.0 29.0 45.0 71.0 64.0 53.0 41.0 28.0 15.0 4.0 0.0		
Field	*Variable*	*Value*	*Description*
1	name	str	Station (basin or outlet) name (KK in *HEC-1*).
2	starttime	+	Military time (0-2400) at the start of the event.
3	interval	+	Interval in minutes between hydrograph ordinates.
4	numordinates	+	The number of hydrograph ordinates.
5-n	flow	+	Hydrograph ordinates. Each of fields 1-5+n are repeated for each station in the file.

The hydrograph file is considered to be free-field when read by *WMS* so any number of spaces can be used to separated the different parameters. Figure 20-40 shows a sample file where there are 25 ordinates with a time interval of 30 minutes.

20.20 Tabular Data File

A tabular data file can be used to import scattered points and their accompanying data sets. These files are imported using the *Import* command in the *File* menu. There are two slightly different formats which allow you to import a set of scatter points with either multiple data sets or a single transient data set. The format of these files is given below:

20.20.1 Tabular Data with Multiple Data Sets

```
"id" "x" "y" "name1" "name2" "name3" …
id x y val1 val2 val3 …
...
...
...
```

Figure 20-41 File Format for Tabular Data with Multiple Data Sets.

```
"id" "x" "y" "2 year" "10 year" "50 year" …
1   3243   48976   .75   1.8   2.25
2   3896   45782   .65   1.3   1.75
3   2439   46346   .82   1.84   2.41
4   2944   49145   .72   1.57   2.3
```

Figure 20-42 Sample Tabular Data File with Multiple Data Sets.

20.20.2 Tabular Data with Transient Data Set

```
"data set name"
     "time1" "time2""time3"   …
id x y val1   val2    val3 …
...
...
...
```

Figure 20-43 File Format for Tabular Data with Multiple Data Sets.

```
"Rainfall"
                    12    12.5  13.0
```

```
1   3243   48976   .10   .21   .08
2   3896   45782   .22   .41   .15
3   2439   46346   .15   .35   .11
4   2944   49145   .18   .21   .05
```

Figure 20-44 Sample Tabular Data File with Multiple Data Sets.

The extra spaces in front of the time values are not important. They are shown here for clarity in reading the file.

20.21 User-Defined Rational Method Dimensionless Hydrograph Files

When running rational method simulations you can use a dimensionless hydrograph to covert the peak flow calculated with the rational method equation into a runoff hydrograph. One of the options is to define your own hydrograph which can be done interactively in *WMS*, or by defining the time and flow values in the file and then importing in the User-defined hydrograph dialog box (see section 0). The format of this fill is shown below:

```
RATIONALTABLE
NUM_ORDS  number_values
TIME_SERIES time-1 time-2 time-3 … time-n
FLOW_SERIES ord-1 ord-2 ord-3 … ord-n
ENDFILE
```

Figure 20-45. Rational Method User-Defined Hydrograph File Format.

```
RATIONALTABLE
NUM_ORDS 12
TIME_SERIES 0  1  2  3  4  5  6  7  8  9  10  11
FLOW_SERIES 0.00  0.20  0.40  0.60  0.80  1.00  0.80  0.60  0.40  0.30
0.20  0.00
ENDFILE
```

Figure 20-46. Sample User-Defined Rational Method Hydrograph File.

Card Type	**RATIONALTABLE**
Description	File type identifier. Must be on first line of file. No fields.
Required	YES

Card Type	**NUM_ORDS**		
Description	Defines the number of time/flow values in the hydrograph table.		
Required	YES		
Format	`NUM_ORDS number_values`		
Sample	`NUM_ORDS 12`		
Field	*Variable*	*Value*	*Description*
1	`number_values`	+	The number of time/flow pairs in the table.

Card Type	**TIME_SERIES**		
Description	Time values for t/t_c.		
Required	YES		
Format	`TIME_SERIES time-1 time-2 time-3 … time-n`		
Sample	`TIME_SERIES 0 1 2 3 5 8 10`		
Field	*Variable*	*Value*	*Description*
1-n	`time-n`	+	ratio of time to time of concentration.

Card Type	**FLOW_SERIES**		
Description	Time values for t/t_p.		
Required	YES		
Format	`FLOW_SERIES time-1 time-2 time-3 … time-n`		
Sample	`FLOW_SERIES 0.0 .25 .67 1.0 .8 .4 0.0`		
Field	*Variable*	*Value*	*Description*
`1-n`	`flow-n`	`+`	ratio of flow to peak flow for time n.

Card Type	**ENDFILE**
Description	End of file identifier. Must be on last line of file. No fields.
Required	YES

20.22 Custom NFF Equations File

Custom NFF equation files are used to defined equations for NFF simulation that are not in the current NFF database. Thus, the latest regression equation can be used in the simulation to ensure the most up to date results possible. Each Custom NFF equation file must be manually created by the user using a text editor. The format of this file may appear formidable at first; this is because it is designed to describe all possible equations. For a simplified explanation of how to create a equation file, please see the end of this section. The format of these files is as follows:

```
CUSTOMNFF                /* File type identifier */
STATE id name            /* State in which equations are valid */
REGIONS numregions       /* Number of hydrologic regions */
BEGREGION name flood_num /* Beg. of region, name, and max flood number
for the region*/
NUMVARIABLES nv          /*Number of variables in equations */
name1 description1 minval1 maxval1 units1 mappingcode1
name2 description2 minval2 maxval2 units2 mappingcodeg2
.
.
.
namei descriptioni minvali maxvali unitsi mappingcodei
EQUATION                   /* Beg. of equation */
STANDARDERROR value        /*Standard error for equation*/
EQUIVALENTYEARS value      /*Equivalent years of record*/
REGRESSIONCONSTANT value   /*Regression constant for equation*/
NUMCOMPONENTS nc           /*number of variables used in equation*/
index mofifier multiplier exp exp_index exp_modifier exp_exp
.
.
.
indexi modifieri multiplieri expi exp_indexi exp_modifieri exp_expi
ENDREGION                 /* End of region */
```

Figure 20-47. Custom NFF Equation File Format.

The following is a sample file for the given information:

State: Wyoming; # of regions: 1

Region 1's equations:

RQ2 = $0.03A^{0.47}*(P-11)^{2.93}$; standard error 100%; No data for equivalent years of record.

RQ5 No Data

RQ10 No Data

RQ25 No Data

RQ50 No Data

RQ100 No Data

RQ500 = $78.4A^{0.52}*(P-11)^{1.84}$; standard error 82%; 7 equivalent years of record.

Variables:

Drainage area (A); minimum=0.10 sq. mi.; maximum=4000 sq. mi.

Mean annual precip. (P); minimum=12.0 in.; maximum=30.0 in.

```
CUSTOMNFF
STATE WY Wyoming
REGIONS 1
BEGREGION Eastern_Region 13
NUMVARIABLES 2
A Drainage_Area 0.10 4000.0 1 0
P Mean_Annual_Precip 12.0 30.0 2 101
EQUATION 2
STANDARDERROR 100.0
EQUIVALENTYEARS 0.0
REGRESSIONCONSTANT 0.03
NUMCOMPONENTS 2
1 0.0 1.0 0.47 0 0.0 0.0
2 -11.0 1.0 2.93 0 0.0 0.0
EQUATION 5
STANDARDERROR 0.0
EQUIVALENTYEARS 0.0
REGRESSIONCONSTANT 0.0
NUMCOMPONENTS 0
.
.
.
EQUATION 500
STANDARDERROR 82.0
EQUIVALENTYEARS 7.0
REGRESSIONCONSTANT 78.4
NUMCOMPONENTS 2
1 0.0 1.0 0.52 0 0.0 0.0
2 -11.0 1.0 1.84 0 0.0 0.0
ENDREGION
```

Figure 20-48. Sample Custom NFF Equation File.

The cards used in Custom NFF Equation file are as follows:

Card Type	**CUSTOMNFF**
Description	File type identifier. Must be on first line of file. No fields.
Required	YES

Card Type	**STATE**
Description	State for which equations will be defined
Required	YES
Format	STATE id name

Sample	STATE WY Wyoming		
Field	*Variable*	*Value*	*Description*
1	id	str	State abbreviation
2	name	str	State name

Card Type	**REGIONS**		
Description	Provides the number of regions for which equations are defined.		
Required	YES		
Format	REGIONS nr		
Sample	REGIONS 3		
Field	*Variable*	*Value*	*Description*
1	nr	1-17	The number of regions. Note: The maximum number of regions that can be defined in one file is 18; if definition for more that 17 regions is needed, separate files must be made and loaded separately.

Card Type	**BEGREGION**		
Description	Marks the beginning of a region definition and gives the name of the region		
Required	YES		
Format	BEGREGION name floodnum		
Sample	BEGREGION Eastern_Region 13		
Field	*Variable*	*Value*	*Description*
1	name	str	The name of the region. If the region is urban, the name must start with the "$" character.
2	floodnum	int	The default maximum U.S. flood region.

Card Type	**NUMVARIABLES**		
Description	Provides the number of variables in the region's equations		
Required	YES		
Format	NUMVARIABLES nv id name minval maxval units mappingflag		
Sample	NUMVARIABLES 2 A Drainage_Area 0.10 4000.0 1 0 P Mean_Annual_Precip 12.0 30.0 2 101		
Field	*Variable*	*Value*	*Description*
1	nv	int	The number of variables in the region's equation
2	id	str	The abbreviated name of the variable.
3	name	str	The name of the variable.
4	minval	+	Minimum value of the variable.
5	maxval	+	Maximum value of the variable.
6	units	0-23	The unit code defining the units used for the variable. See below for the table of unit codes.
7	name	str	Flag to indicate whether or not the variable can be mapped by *WMS*. See mapping code table below.

Card Type	**EQUATION**
Description	Marks the beginning of an equation definition.
Required	YES

Format	EQUATION num		
Sample	EQUATION 100		
Field	*Variable*	*Value*	*Description*
1	num	int	The flood interval corresponding to the equation. Note: An equation card for each interval: 2, 5, 10, 25, 50, 100, 500 years must be defined in the file.

Card Type	**STANDARDERROR**		
Description	Provides the standard error given for the equation.		
Required	YES		
Format	STANDARDERROR value		
Sample	STANDARDERROR 85.3		
Field	*Variable*	*Value*	*Description*
1	value	+	The standard error for the equation. 0.0 is entered if no standard error data exists.

Card Type	**EQUIVALENTYEARS**		
Description	Provides the number of equivalent years for the equation.		
Required	YES		
Format	EQUIVALENTYEARS value		
Sample	EQUIVALENTYEARS 12.0		
Field	*Variable*	*Value*	*Description*
1	value	+	The equivalent number of years record for the equation. 0.0 is entered if no data exists.

Card Type	**REGRESSIONCONSTANT**		
Description	Provides the regression constant of the equation.		
Required	YES		
Format	REGRESSIONCONSTANT value		
Sample	REGRESSIONCONSTANT 0.03		
Field	*Variable*	*Value*	*Description*
1	value	+	The constant term of regression for the equation. 0.0 is entered if no data exists.

Card Type	**NUMCOMPONENTS**
Description	Gives the number of components (excluding constant) in the regression equation.
Required	YES
Format	NUMCOMPONENTS nc index modifier multiplier exp expindex exp_modifier exp_exp

Sample	NUMCOMPONENTS 2 1 0.0 1.0 0.47 0 0.0 0.0 2 -11.0 1.0 2.93 0 0.0 0.0		
Field	*Variable*	*Value*	*Description*
1	nc	int	The number of components in the regression equation. 0.0 is entered if no data exists.
2	index	int	Base variable index. The index of the base variable within the list of name and descriptors entered above. Example: 1 (for A); 2 (for P). 0.0 is entered if no variable exists.
3	modifier	+	Modifier added to base variable before the multiplier is applied. 0.0 is default.
4	multiplier	+	Value by which base variable is multiplied. 1.0 is the default value.
5	exp	+	The exponent constant for the base variable. 1.0 is the default.
6	expindex	int	The index of the variable which appears in the base variable exponent. 0.0 is the default.
7	exp_modifier	+	Modifier added to exponent variable. 0.0 is default.
8	exp_exp	+	The exponent constant for the exponent variable. 1.0 is the default.

Card Type	**ENDREGION**
Description	Marks the end of a region definition.
Required	YES

The units codes are given in the following table:

Code	*English Units*	*SI units*	*Conversion Factor*
0	no units	no units	1.0000
1	mi^2	km^2	2.589988
2	inches	mm	25.400
3	Fahrenheit	Celcius	**Formula**.
4	1000*ft	km	0.30480
5	ft/mile	m/m	0.000189
6	mile	km	1.609344
7	feet	meter	0.304800
8	in/hr	mm/hr	2.54000
9	mi/mi	km/km	1.0000
10	%	%	1.0000
11	°arc	°arc	1.0000
12	area ratio	area ratio	1.0000
13	decimal fraction	decimal fraction	1.0000
14	cfs	m^3/s	0.028317
15	inches(WI)	mm(WI)	25.400
16	ft/ft		1.0000
17	mm/yr		25.400
18	mm/yr		0.386102
19	strm/mi^2	strm/km^2	0.386102
20	mi/mi^2	km/km^2	0.621371
21	person/mi^2	person/km^2	0.386102
22	decimal°	decimal°	1.0000
23	mi^2/mi^2	km^2/km^2	1.0000

Unit Codes Table

Mapping codes for variables are given in the following table:

Code	*Variable*
0	Area
1	Shape factor
2	Elevation
3	Channel slope
4	Channel length
5	Altitude index
6	Relief
7	Effective Drainage
8	Drainage area below 8000 ft elevation
9	Sinuosity
10	Basin above 6000 ft elevation
100	Any other variable not included in 1-10

Mapping Codes Table

20.22.1 Simplified explanation of creating a custom equation file:

The majority of NFF equations have only a handful of basic components; many of the values in the above cards are 0 or 1. Commonly an equation will have simply a constant of regression, followed by a few variables raised to some exponential power such as the following equation from Mississippi:

$$\mathbf{Q2 = 66.2A^{0.88}S^{0.51}L^{-0.11}}$$

To define a file for this equation, follow these steps (the lines in bold indicate lines to be included in the file):

1) The first line of a file is always: **CUSTOMNFF**

2) The next line (for Mississippi) is : **STATE MS Mississippi**

3) Assume (for simplicity) Mississippi has only 1 region. The next line in the file would be: **REGIONS 1**

4) Suppose that the above equation is given for Region 1 . The steps to define a region's equations are:

   a) Start the region with: **BEGREGION Region_1 3**. The last number, 3, is the U.S. maximum flood region in which Mississippi is located (refer to the NFF documentation if you are unsure which maximum flood region you are in).

   b) For the next line, count the number of variables used in the equations for the region. In this case, 3 (A, S, L): **NUMVARIABLES 3**.

c) The lines under this card are the definition of each variable A, S, and L. For each variable, a line in the file must contain the abbreviated name, name, minimum value, maximum value, units code and mapping code, in that order (see NUMVARIABLES card explanation above). The units code and mapping code for each variable is found in the tables above. For this region's equations, these lines are:

A Drainage_area 0.10 4000.0 1 0

S Mean_channel_slope 0.0003 0.001 5 3

L Channel_length 1.0 500.0 6 4

d) Next, each equation is defined specifically. The following cards must be included for all 7 equations (Q2--Q500) of a region whether they are defined of not. Suppose that in this region, the Q500 equation is not defined. The equations would be listed in the file as follows:

EQUATION 2 /* Recurrence interval */

STANDARDERROR 45.0

EQUIVALENTYEARS 0.0

REGRESSIONCONSTANT 66.2

NUMCOMPONENTS 3 /*Number of variables */

Under the NUMCOMPONENTS card, the coefficients and exponents of each variable is defined. Each variable is defined by index, modifier, multiplier, exponent, exponent index, exponent modifier, exponent exponent (for an explanation of each of these, see the NUMCOMPONENTS card table above). For the majority of NFF equations only the index and exponent of each variable is different. The rest of these fields are 0 except the modifier which is 1.0. For the above Q2 equation the file lines are:

0.0 1.0 0.88 0 0.0 0.0

0.0 1.0 0.51 0 0.0 0.0

0.0 1.0 -0.11 0 0.0 0.0

All these cards must be repeated for all 7 intervals; if the equation of a certain interval doesn't exist, such as Q500 in this case, the cards are as follows:

EQUATION 500 /* Recurrence interval */

STANDARDERROR 0.0

EQUIVALENTYEARS 0.0

REGRESSIONCONSTANT 0.0

NUMCOMPONENTS 0 /*Number of variables */

Component definition cards are not needed since the number of components is 0.

e) The region definition is then ended by: **ENDREGION**

f) This procedure is repeated for each region of the state (if more than one is given).

The file is now ready to be used by *WMS*. Save it as customnff.equ in the same directory where the regular NFF database is stored; *WMS* will then be able to load the file automatically when "Custom" is selected in the State dialogs of NFF.

20.23 XY Series Files

The *XY Series Editor* described in Appendix B is used in several places in *WMS*. The *XY Series Editor* is a general purpose editor for entering curves or pairs of lists of data. The *XY Series Editor* allows a curve to be imported from a file, created and edited graphically, or created and edited using two columns of edit fields in a spreadsheet-like interface.

XY series files can be used to prepare a set of curves for import to the *XY Series Editor*. XY series files are also used to export curves generated within the Editor for future use.

The format of the XY series file is shown in Figure 20-49, and a sample file is shown in Figure 20-50. Curves are defined in an XY Series File using one of three types of cards: XY1, XY2, or XY3. With the XY1 card, both the x and y values are listed for each point on the curve. There is no limit to the spacing or interval used between subsequent x values. The XY2 card is identical to the XY1 card except that the number of points and the x values are assumed to be static and cannot be altered by the user. With the XY3 card, the x values are defined by a beginning x value, an initial increment in x, and a per cent change in x per increment. Only the y values are explicitly listed.

```
XY1 id n dx dy rep begc name             /* XY Series vers. #1 */
x1 y1                                    /* XY values */
x2 y2
.
.
xn yn
XY2 id n dx dy rep begc name             /* XY Series vers. #2 */
x1 y1                                    /* XY values */
x2 y2
.
.
xn yn
XY3 id n x1 incx pcx dx dy rep begc name /* XY Series vers. #2 */
y1                                       /* Y values */
```

```
y2
.
.
yn
```

Figure 20-49. The XY Series File Format.

```
XY3 1 241 0 6 0 0 0 0 0 typeI-24hour
0.00000
0.00174
.
.
1.0000
```

Figure 20-50. The Sample XY Series File.

The card types used in the XY series file format are as follows:

Card Type	**XY1**		
Description	Defines a curve with a list of XY values. Any number of points and any x spacing between points may be used.		
Required	NO		

Format	`XY1 id n dx dy rep begc name`
	`x1 y1`
	`x2 y2`
	`.`
	`.`
	`xn yn`		

Sample	`XY1 1 5 0 0 0 0 head`
	`0.0 0.0`
	`1.0 2.0`
	`2.5 7.0`
	`3.0 8.0`
	`4.5 9.5`		

Field	*Variable*	*Value*	*Description*
1	id	+	The id of the XY series.
2	n	+	The number of point in the series.
3	dx	0,1	A flag defining whether the x values listed are to be interpreted as incremental (dx=1) or absolute (dx=0).
4	dy	0,1	A flag defining whether the y values listed are to be interpreted as incremental (dy=1) or absolute (dy=0).
5	rep	0,1	A flag defining whether the xy series is to be interpreted as cyclic (repeating)
6	begc	±	The x value in the series where the cyclic portion of the curve begins. Value is ignored if rep=0.
7	name	str	The name of the series.
8-9	x,y	±	The xy values of the points defining the curve. Repeat n times.

Card Type	**XY2**
Description	Defines a curve with a list of XY values. This card is identical to the XY1 card except that the number of points and the x values are assumed to be static and cannot be altered by the user.

Card Type	**XY3**		
Description	Defines a curve with a list of Y values. The x values are defined by a beginning value, an increment, and a bias.		
Required	NO		

Format	`XY3 id n x1 incx biasx dx dy rep begc name`
	`y1`
	`y2`
	`.`
	`.`
	`yn`		

Sample	`XY3 1 10 0 1 0 0 0 0 0 head`
	`0.0`
	`2.0`
	`7.0`
	`8.0`
	`9.5`
	`9.1`		

Field	*Variable*	*Value*	*Description*
1	id	+	The id of the XY series.
2	n	+	The number of point in the series.
3	x1	±	The first x value.
4	incx	±	The increment in x used to compute the next x value.
5	pcx	+	The per cent change in x used to compute subsequent x values. Expressed as a decimal, i.e., 0.05 = 5%.
6	dx	0,1	A flag defining whether the x values listed are to be interpreted as incremental (dx=1) or absolute (dx=0).
7	dy	0,1	A flag defining whether the y values listed are to be interpreted as incremental (dy=1) or absolute (dy=0).
8	rep	0,1	A flag defining whether the xy series is to be interpreted as cyclic (repeating)
9	begc	±	The x value in the series where the cyclic portion of the curve begins. Value is ignored if rep=0.
10	name	str	The name of the series.
11	y	±	The y values of the points defining the curve. Repeat n times.

CHAPTER 21

The XY Series Editor

21.1 Introduction

The *XY Series Editor* is a special dialog that is used to generate and edit curves defined by a list of x and y coordinates. The curve can be created and edited by directly editing the xy coordinates using a spreadsheet-like list of the coordinates. The curve can also be generated and edited graphically. An entire list of curves can be generated and edited with the Editor and curves can be imported from and exported to text files for future use.

The *XY Series Editor* is used in several places in *WMS*. It was designed to be general in nature so that it could be used anywhere that a curve or function needs to be defined. In some cases, the x values of the curve must correspond to a pre-defined set of values. For example, the x values may correspond to a set of time steps whose interval is established in a separate dialog. In such cases, the x fields cannot be edited but the y values associated with the pre-defined x values can be edited. In other cases, there is no limit on the number of x values or on the x spacing and both the x and y values can be edited. The different types of series used by *WMS* are listed below:

- **Rainfall Series** - Have fixed X values representing time and Y values representing precipitation. These series are used to define PI and PC cards from both the gage and precipitation dialogs.

- **Hypothetical Storm Series** - Have fixed X values which correspond to the times for the pH card. The Y values represent the rainfall accumulation at the specified time.

- **Time Area Series** - Have fixed X values representing time and Y values representing area. These series are used to define time area distribution curves for the UA card.

- **Storage Routing Series** - Have fixed X values representing the consecutive number of each Y value. The Y values can represent elevations, areas, flows, or volumes, depending on which options are selected.

- **Cross Section Series** - Have X and Y values representing the cross-section points of a channel.

- **Diversion Series** - Have fixed X values, and Y values representing flow rates for the corresponding DI and DQ records.

- **Unit Hydrograph Series** - Have fixed values representing the time values as defined by the IT record. Y values need to be entered and represent the unit hydrograph ordinates.

- **Given Hydrograph Series** - Have fixed values representing the time values as defined by the IN card in the basin parameters dialog.

The *XY Series Editor* is shown in Figure B.1. Each component of the dialog is described below.

21.2 XY Series List

At the bottom of the dialog in the center, there is a list of xy series. One of the items in the list is active and highlighted at all times. The xy values of the active series are shown in the spreadsheet on the left side of the dialog and the curve is shown graphically in the upper right portion of the dialog. The name associated with the active series can be edited using the edit field to the right of the xy series list.

A new xy series can be created and added to the xy series list by selecting the *New* button. An existing series can be copied to create a new series by selecting the *Duplicate* button. This option is useful when two series need to be the same except for slight differences. An existing series can be deleted from the list by highlighting the series and selecting the *Delete* button to the right of the xy series list. A set of series can be read from a file by selecting the *Import* button. Likewise, the entire list of series can be saved to a file using the *Export* button. The file format used to save xy series is described in Appendix A.

21.3 XY Edit Fields

The two vertical columns of edit fields on the left side of the dialog are for direct editing of the xy series values. A pair of application specific titles appears at the top of the columns. The *TAB* key can be used to move the cursor through the edit fields.

If the number of points in the series is greater than the number of pairs of fields in the columns, the scroll bar to the right of the columns can be used to scroll through the entire range of the xy series. If the x fields are dynamic (can be edited), a new point can be created at the end of the series by moving the cursor to the last field and hitting the *TAB* key.

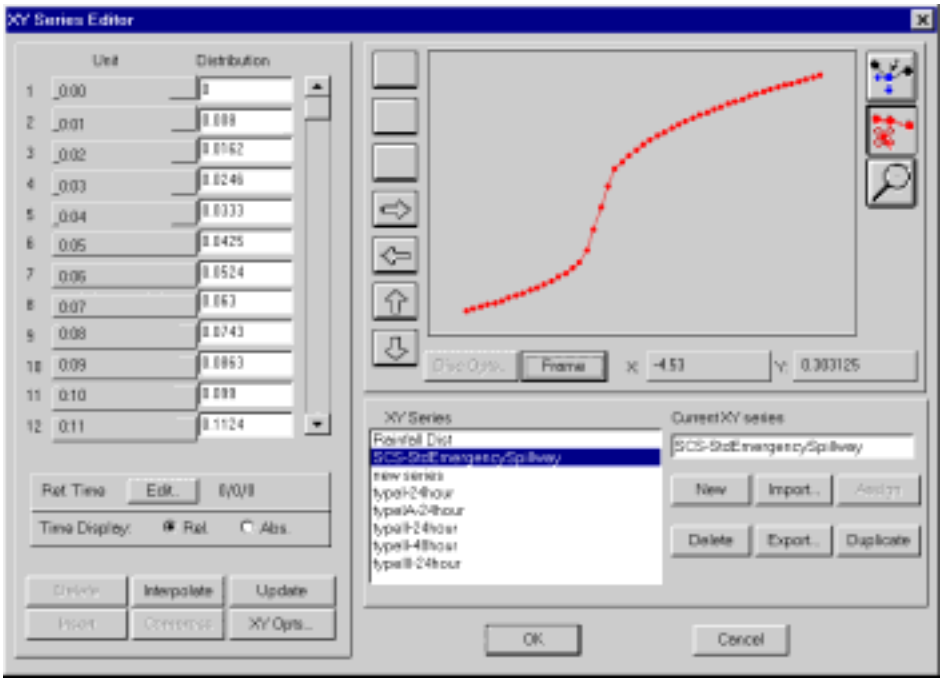


Figure 21-1. The XY Series Editor.

The buttons below the xy edit fields are used to manipulate the values in the edit fields. The buttons are as follows:

21.3.1 Delete

The *Delete* button blanks (clears) the edit field that the cursor is located in. If the x field is static, only the y field is cleared. Otherwise, both fields are blanked.

21.3.2 Interpolate

The *Interpolate* button causes any blank fields in the xy series to be filled in by linear interpolation between the closest non-blank fields above and below the blank fields.

21.3.3 Update

The *Update* button redraws the xy series curve in the plot window using the current values in the edit fields.

21.3.4 Insert

The *Insert* button adds a new point to the xy series by adding a pair of blank fields just above the field containing the cursor.

21.3.5 Compress

The *Compress* button reduces the length of the xy series by removing all points whose edit fields are blank.

21.4 XY Series Plot

The window in the upper right hand corner of the *XY Series Editor* is used to plot the curve corresponding to the xy values in the edit fields. As each value in the edit fields is edited, the corresponding point on the curve is adjusted instantaneously. The plot provides an immediate visual feedback to the user which is helpful in detecting erroneous values on input.

21.4.1 The Plot Tools

The *Plot Window* can also be used to edit the xy series graphically. The following tools (found on the right side of the plot) are used for graphical editing:

Select Point Tool

The *Select Point* tool is used to graphically change the xy values of a point by clicking on the point with the cursor and repositioning the point while holding down the mouse button. The tool can also be used to select points for deletion. A set of points can be selected by clicking on the points with the SHIFT key depressed or by dragging a box around a set of points. The selected points can then be deleted by selecting the Delete button beneath the xy edit fields.

Create Point Tool

The *Create Point* tool is used to graphically add new points to a curve by clicking in the plot window at the location of the new point.

Zoom Tool

The *Zoom* tool is used to zoom in on a region of the curve being plotted in the plot window. Clicking on a point zooms the view by a factor of two around the point. Dragging a rectangle alters the mapping so that the region in the rectangle fills the plot window. Holding the *SHIFT* key down while clicking in the plot window causes the view to be enlarged by a factor of two around the point clicked.

21.4.2 The Plot Macros

The buttons to the lower left of the plot window in the *XY Series Editor* are used to pan the view in plot window up, down, left, or right. After altering the view using either the *Pan* buttons or the *Zoom* tool, the curve can be centered in the plot window by selecting the *Frame* button beneath the plot window.

The buttons to the upper right of the plot window are to quickly create curves using analytic functions. Each button brings up a dialog that allows the parameters of a function (ex. sine curve) to be specified from which a series of xy points are created.

21.5 Defining A Rainfall Series (PI/PC)

As mentioned previously, PI and PC cards are defined from the gage and precipitation dialogs. If the XY Series Editor is invoked from either of these dialogs, any rainfall series already defined in this session will be displayed in the list box. If it is the first time the editor has been used in this session or if a series has not been assigned to the given gage/precipitation record, an empty, default series will be active and ready for definition. If the gage/precipitation record already has a series assigned to it, that series will be the current or active series when the editor is invoked.

For all rainfall series, the X field represents time and will be fixed. The starting time and interval can be set using the XY options as described above. The Y field represents either the incremental or the cumulative rainfall for the PI or PC card. **The default is cumulative (PC) so that if you wish to enter incremental or delta values, you must use the XY Options to set the Y values to delta instead of absolute.** A maximum of 300 values are allowed for each rainfall series. The XY Series Editor will not permit more values than this to be entered for a rainfall series.

When the OK button is selected, the current series is assigned to the gage or precipitation record (depending on the dialog from which the editor was invoked). Therefore, to assign a previously defined series to a second, third, etc. gage or precipitation record, select the desired series from the list box once the editor has been invoked and select the OK button. In the same way, series created from a previous session of *WMS* can be imported using the *Import* option.

If PI or PC cards are encountered while reading an *HEC-1* input file, a series is added to the rainfall series list at the same time it is associated with the appropriate gage or precipitation record.

21.6 Defining A Hypothetical Storm Series (pH)

The XY Series Editor is used to input values for a hypothetical storm (pH) card. If the XY Series Editor is invoked from this dialog, any hypothetical storm series already defined in this session will be displayed in the list box. If it is the first time the editor has been used in this session or if a series has not been assigned to the given hypothetical storm record, an empty, default series will be active and ready for definition. If the hypothetical storm record already has a series assigned to it, that series will be the current or active series when the editor is invoked.

In this case, fill in the rainfall accumulation for the specified times using the Y edit fields.

21.7 Defining A Time Area Curve (UA)

A time area curve must be defined when the time area method for hydrograph computation is selected in the *Unit Hydrograph* dialog. If the XY Series Editor is invoked from this dialog, any time area series already defined in this session will be displayed in the list box. If it is the first time the editor has been used in this session or if a series has not been assigned to the given time area record, an empty, default series will be active and ready for definition. If the time area record already has a series assigned to it, that series will be the current or active series when the editor is invoked.

The X field represents the time and the Y field represents the percentage of the basin area contributing to runoff at the given time. The starting time and interval is determined from the Job Control parameters and can be adjusted using the starting time and interval specified in the *XY Options*. Up to 150 values can be assigned. However, not all of the area (Y) fields need to be defined. *WMS* will only write out those fields for which a value has been specified.

21.8 Defining A Storage Routing Curve (RS)

Storage routing curves must be defined for the SE, SV, and SQ options of the storage routing option. The XY Series Editor is invoked whenever the *Define* button is selected from a dialog used to define storage routing. Any time area series already defined in this session will be displayed in the list box. If it is the first time the editor has been used in this session or if a series has not been assigned to the given storage routing record, an empty, default series will be active and ready for definition. If the storage routing record already has a series assigned to it, that series will be the current or active series when the editor is invoked.

The X field is simply a cumulative numbering of the values entered and the Y field represents either the elevations, volumes, or outflows, corresponding to the SE, SV and SQ cards, depending upon the options set when the XY Series Editor was invoked. The label above the Y series values will serve as a reminder as to the type of values the XY Series Editor is expecting. Up to 20 values may be entered for each of the storage routing options and the editor will not allow more values to be entered for a storage routing curve. The graphical display can be used as a visual check of the validity of the values entered for the storage routing curves.

21.9 Defining A Cross-Section (RX/RY)

Cross-sections can be used with storage routing when the normal depth option is chosen, as well as in conjunction with Muskingum-Cunge routing. Before actually entering the cross-section the Manning coefficients for the channel, left overbank ,and right overbank as well as the length and slope must be entered for the RC record. From the dialog used to enter the RC record values the XY series editor can be invoked. Unlike the other series, both the X and Y values can be edited. X values are used as the horizontal stations on the RX record, and Y values define the vertical elevations on the RY record.

21.10 Defining Diversion Data (DI/DQ)

Stream flow diversions are defined on DI/DQ records where the DI record describes flow rates upstream of the diversion whereas the DQ record describes the corresponding diversion rates. Both are defined using the XY series editor where the Y values represent the flow rates at a given level. The same number of flow rates should be defined on the DI and DQ records.

21.11 Defining A Unit Hydrograph (UI)

One of *HEC-1*'s unit hydrograph options is to simply provide a computed unit hydrograph. Unit hydrographs are entered using the XY series where the X

fields are fixed as time increments of the unit hydrograph as specified on the IT record (defined using the *Job Control* command). Y values represent the unit hydrograph ordinates. Do not change the starting time and time intervals as with other series, because the unit hydrograph time parameters should be identical to the computational time parameters. The XY series editor is invoked from within the *Unit Hydrograph Method* dialog when the UI option is selected.

21.12 Defining A Given Hydrograph (QI)

HEC-1 allows a measured or given hydrograph to be used for a basin. This can be done by specifying the given hydrograph toggle in the basin parameters dialog and then defining the hydrograph by invoking the XY Series Editor from within the same dialog. Like the unit hydrograph series, the X fields are fixed as time increments of the given hydrograph, and Y values represent the hydrograph ordinates. However the time parameters are set by changing the starting time and increment using the XY options from within the XY Series Editor. The day, month, and year used to complete the IN record values preceding the QI record are defined prior to invoking the XY Series Editor.

REFERENCES

Anonymous, 1963, The application of synthetic unit hydrographs to drainage basins in the Riverside County Flood Control and Water Conservation District, Riverside, CA.

Clark, C.O., 1945, Storage and the unit hydrograph, *Transactions of the American Society of Civil Engineers,* vol. 110, pp. 1419-1446.

Clough, R. W., and J. L. Tocher, 1965, Finite element stiffness matrices for analysis of plates in bending, *Proc. Conf. Matrix Methods in Structural Mechanics, Wright-Patterson A.F.B., Ohio, Air Force Flight Dynamics Lab., Research and Technology Division, Air Force Systems Command, The Air Force Institute of Technology, Air University,* pp. 515-545.

Dodson & Associates Inc., 1992, The Dodson Professional *HEC-1* System, PRO*HEC-1* Documentation.

Eagleson, P. S., 1962, Unit hydrograph characteristics of sewered areas, *Proceedings of the ASCE, HY2, American Society of Civil Engineers*.

Espey, Huston & Associates Inc., P. S., 1987, Drainage criteria Manual for Fort Bend County, Texas, *Prepared for the Fort Bend County Drainage District*.

Espey, W.H. Jr., W. Howard Jr., C.W. Morgan, and F.D. Marsch, 1966, Study of some effects of urbanization on storm runoff from a small watershed, *Texas Water Development Board,* Report 23.

Espey, W.H. Jr., D.G. Altman., and C.B. Graves Jr., 1977, Nomographs for ten-minute hydrographs for small urban watersheds, *ASCE Urban Resources Research Program,* Technical Memorandum No. 32.

Franke, R. & G. Nielson, 1980, "Smooth interpolation of large sets of scattered data," *International Journal for Numerical Methods in Engineering,* Vol. 15, pp. 1691-1704.

Garbrecht J., and L.W. Martz, 1995, "TOPAZ: An Automated Digital Landscape Analysis Tool for Topographic Evaluation, Drainage Identification, Watershed Segmentation, and Subcatchment Parameterization: Overview," *U.S. Department of Agriculture, Agricultural Research Service,* ARS Publication No. NAWQL 95-1, 17 pp., October 1995.

Jones, N. L., 1990, *Solid Modeling of Earth Masses for Applications in Geotechnical Engineering,* Ph.D. Dissertation, The University of Texas at Austin, 324 p.

HEC, 1990, *HEC-1 Flood Hydrograph Package*, User's Manual, U.S. Army Corps. of Engineers Hydrologic Engineering Center.

Kerby, W.S., 1959, Time of concentration for overland flow, *Civil Engineering.* Vol. 29, No. 3, p. 174.

Kirpich, Z.P., 1940, Time of concentration of small agricultural watersheds, *Civil Engineering.* Vol. 10, No. 6, p. 362.

Nelson, E.J., *Automated Watershed Characterization Using Triangulated Irregular Networks*, Ph.D. Dissertation, Brigham Young University, April 1994, 128 pp.

Nelson, E.J., N.L. Jones, and A.W. Miller, 1994, "An algorithm for precise drainage basin delineation", *ASCE Journal of Hydraulic Engineering*, Vol. 120, No. 3, pp. 298-312.

Puecker, T.K., and D.H. Douglas, 1975, "Detection of surface-specific points by local parallel processing of discrete terrain elevation data," *Computer Graphics and Image Processing*, Vol 4. pp. 375-387

Putnam, A.L., 1972, Effect of urban development on floods in the Piedmont Province of North Carolina, *U.S. Geologic Survey Open File Report*, Washington, D.C.

Ramser, C.E., 1927, Runoff from small agricultural watersheds, *Journal of Agricultural Research*, Vol. 34, No. 9, pp. 797-823.

Shepard, D., 1968, "A two dimensional interpolation function for irregularly spaced data," *Proc. 23rd National Conference of the ACM*, pp. 517-523.

Soil Conservation Service (SCS), 1972, Hydrology, *National Engineering Handbook, U.S. Department of Agricultural*, Section 4.

Taylor, A.B., and H.E. Schwartz, 1952, Unit hydrograph lag and peak flow related to basin characteristics, *Transactions of the American Geophysical Union*, Vol. 33, pp. 235-246.

Wright-McLaughlin Engineers, 1975, Urban storm drainage criteria manual, *Denver Regional Council of Governments*, Volumes I and II.

Index

2D grid
boundary....17-5
bounding 2D scatter point set....18-19
cell....17-2
cell-centered....17-1, 17-6
contour labels....17-3
contouring....17-5
conversion to other data types....17-7
creating....2-18, 17-5
data set....20-15
display options....17-4
file....20-8
fringing....17-5
gage....17-3, 17-4
import....2-18
mesh-centered....17-1, 17-6
types....17-1
2D Grid
CASC2D....19-4
2D scatter point
active....18-1
bounding grid....18-19
data set....18-1
display options....18-3
editing....18-2
file....16-3, 18-2, 20-10
from 2D grid....17-7
icon....18-2
numbers....18-3
reading from a file....18-2
saving to a file....18-2
selecting....18-2
set....18-2
symbols....18-3
3D grid
data set....20-15
abort
drawing....2-34
activate DEM points....4-5
active
2D scatter point set....18-1, 18-2
cell....17-6
data set....18-4
gage....16-9
xy series....21-2
active/inactive
activate with polygon....17-7
animation....16-4, 16-6–16-9
transient....16-8
arc
create....3-8, 5-3
arc type
CASC2D....3-23
drainage....3-12
arc vertices
edit....3-7, 5-2
ARC/INFO
ASCII grid file....20-20
export....2-21
shape files....2-10
arcs....3-4
creating polygons....3-33
creating streams from....3-33
dangling....3-27
intersecting....3-27
reverse direction....3-33
time computation....15-25
arcview superfile....2-14
area....10-12, 11-7, 15-12
computing for basins....5-11, 7-12
elevation zones....7-13
aspect....15-12
assign land use....3-16
assign soil type....3-17
attributes
DEM....2-18
auto redraw....2-35
auto-center....2-36
tree....9-3
xy series....21-4
BACKSPACE key
deleting....2-26
select by polygon....2-27
selecting vertex strings....6-2
barycentric....18-11
base flow
TR-20....11-4
base flow....10-13
basin
delineation commands....7-1
basin delineation....5-10
basins

area 5-11, 7-12
attribute display color 7-5
attributes
area 15-12
aspect 15-12
average overland flow distance 15-12
distance from centroid to stream 15-13
maximum flow path 15-12
maximum stream length 15-13
slope 15-12
stream lenghts 15-13
stream slope 15-13
centroids 7-4
computing attributes 5-11, 7-12
creating in trees 9-5
defining 7-10
definition of 7-10
deleting 7-11
deleting from trees 9-5
deleting triangles outside of 7-11
displaying 7-4
exporting attributes 2-21
geometric parameters 6-3
id 7-4
labels 6-3
merging 7-11
names 7-4
outlining 7-4
refined boundaries 7-10
select 9-2
splitting 7-11
tree 9-1
bearing 2-36
beginning time 10-4
bias 17-6
boundary of TIN 6-5
breakline 7-7
options 6-16
TIN 6-2, 6-16
calculator
channel calculations 15-34
culverts 15-40
detention basins 15-36
weirs 15-35
calibration 9-8, 16-9
card 20-1
CASC2D
arc type 3-23
availability 1-2
Channel Routing 19-7
coverages 3-22
creating grids 19-4
defining channel properties 19-15
editng maps 19-14
executing from WMS 19-15
introduction 19-1
job control 19-6
links and nodes 3-38
map editing 19-10
map file 20-20
model checker 19-15
Output Control 19-8
overland flow 19-12
point/node type 3-22
polygon type 3-24
post processing 19-16
precipitation 19-8
project file 19-2
reading data 19-4
reclassification 19-11
saving data 19-3
smoothing stream cell elevations 3-37
soils 20-21
steps for model creation 19-1
tool palette 19-2
using feature objects with 3-37
cell
active 17-5, 17-6, 17-7
creating 17-6
displaying 17-5
inactive 17-3, 17-5, 17-6
selecting 17-2
channel calculations 15-34
channel edges 6-6, 7-7
channel losses 10-29
circumcircle 6-6, 6-12
Clark storage coefficient 15-23
Clark unit hydrograph 10-18
clean
feature objects 3-27
clipboard 2-27
Clough-Tocher ..... (see interpolation)
CN
computation step 15-4
computing 15-1
displaying 7-5
land use 3-16
soil type 3-17
color
arcs 3-25
background 2-4
basin attribute 7-5
cell 17-5
DEM 4-2, 5-4
flow path 6-4
grid node 17-4

HSV 2-32
intensity 2-32
ramp 16-5
TIN 6-4
watershed 7-5
combined hydrographs 10-10, 10-23
comments in HEC-1 10-10
composite curve number table 15-5
computation step 15-4
compute
time of concentration 15-24
compute basin attributes 5-11, 7-12
computing
lag time 15-5
time of concentration 15-5
computing flow directions 5-12
computing gage weights 10-14
confirm deletions 2-27
confirm z value 6-10
confluences 7-6
contour 2-30, 16-5
2D grid 17-5
animation 16-8
colors 2-32
displaying 6-5
flood plains 8-8
label tool 2-33
labels 2-33, 4-2, 6-3, 17-3
legend 2-31
logarithmic scale 2-31
maximum 2-31
minimum 2-31
options 2-31
spline 2-31
TIN 6-14
values 2-31
converting DEM basins to polygons 5-11
convex hull 6-13, 6-17
coordinates 2-7
barycentric 18-11
gage 16-11, 17-3
local 18-16
snapping to a grid 2-35
vertex 6-1
xy series 21-1
copy 2-27
copyright i
coverage
CASC2D 3-22
cross section 3-19
drainage type 3-11
general 3-22
land use 3-14
MODRAT 3-19
runoff coefficient 3-18
soil type 3-16
time computation 3-18
type 3-9
visibility 3-6
coverage box 2-6
create streams 7-7
creating watershed models in WMS 1-3
critical interior points 6-22
cross section editor 3-20
cross sections 21-7
CTRL key
selecting triangles 6-2, 6-17
cumulative rainfall 21-5
curve number 11-7
custom nff equation
file 20-34
D8 5-7
data calculator 16-4
data set
2D grid 20-15
3D grid 20-15
active 16-3, 18-4
animation 16-6–16-9
browser 16-2
calculator 16-4, 16-5
contouring 16-5
deleting 16-4
elevation 16-4
export 16-3
file 16-3
file (ASCII) 20-12
file (binary) 20-16
fringing 16-5
import 16-3
info 16-4
legend 2-31
maximum 16-4
mean 16-4
minimum 16-4
name 16-4
standard deviation 16-4
statistics 16-4
time step 16-3, 16-9, 18-4
data type conversion
2D grid->2D scatter points 17-7
2D grid->TIN 17-8
TIN->scatter points 6-24
defaults 2-10
degree-day 10-21
Delauney ..... (see triangulation)
delete 2-5

all data 2-8
all data of a module..... 2-27
basins from trees 9-5
command 2-26
confirm.....2-27, 3-35, 6-9
data set 16-4
duplicates..... 6-12
feature objects..... 3-35
flood plain..... 8-10
gage..... 16-11
hydrograph..... 9-8
images..... 3-46
Inactive DEM points..... 4-6
outlets from trees 9-5
reservoir..... 7-10
reservoirs from trees 9-5, 9-6
streams..... 7-8
triangles 6-2
vertices..... 6-1, 6-9
xy series 21-3
DELETE key
deleting 2-26
select by polygon 2-27
delineate flood plain..... 8-8
DEM
activate points..... 4-5
active region 4-5
attributes..... 2-18
basins 5-10
basins to polygos..... 5-11
clipping..... 2-17
computing flow directions 5-12
converting streams to feature objects..... 5-9
creating grids from..... 3-35
creating TINs from 3-33
delete portions 4-6
display options..... 5-4
displaying contours..... 4-3
displaying points..... 4-3
drawing flow paths..... 5-2, 5-11
file..... 2-8, 20-5
filter 4-7
flow accumulations 5-7
flow directions 5-7
importing attributes 2-18
importing files..... 2-15
inactivate points..... 4-6
inactive points..... 5-11
land use id's..... 4-4
memory requirements 2-17
module 1-24
polygon basin ids 5-12
shading..... 2-34
smoothing elevations..... 4-6
soil type 4-4
status..... 4-5
thinning 2-17
watershed delineation 1-6, 5-1
where to download from internet..... 1-7
DEM
displaying 4-4
displaying flat cells..... 4-4
demo version..... 2-25
detention basin calculations 15-36
dew point 10-23
digital line graph..... 2-14
Digital Terrain Modeling..... 1-2
digitizing streams..... 20-25
dimensionless hydrograph
file 20-33
dip..... 2-36
display options
2D grid 17-4
2D scatter point 18-3
DEM..... 4-2, 5-4
drainage..... 7-2
dxf files..... 3-47
feature objects 3-25
flood..... 8-2
gage curve 16-13
gages..... 16-10
grid 2-35
hydrograph 9-7
images 3-43
printing..... 2-22, 2-24
TIN..... 6-4
tree..... 9-2
vertices 6-10
displaying flow paths 7-5
diversion
creating..... 10-30
defininng TR-20 11-14
edit..... 10-31
select..... 9-2
diversions
adding to trees 9-5
retrieving 9-6
DLG files..... 2-14
dragging
to select triangles..... 6-17
drainage
arc type..... 3-12
display options 7-2
point/node type..... 3-12
polygon type..... 3-13
drainage analysis with DEMs 5-9

drainage distance....6-4
drainage slope....6-4
drawing depth....3-41
drawing grid
creating points....3-8
drawing objects....3-39
create ellipse....3-39
create line....3-39
create rectangle....3-39
create text....3-39
drawing depth....3-41
editing....3-40
line attributes....3-41
move to back....3-42
move to front....3-42
moving....3-40
rectangle and oval attributes....3-41
selecting....3-40
shuffle down....3-42
shuffle up....3-42
text attributes....3-40
tools....3-39
duplicate vertices....6-12
DXF
export....2-22
import....2-20
dxf files....3-47
converting to feature objects....3-48
converting to TINs....3-49
deleting objects....3-48
display options....3-47
importing....3-48
edit
diversion....10-31
gage....10-32
stage....8-5
TIN....6-3
TINs automatically....6-18
edit box....2-6
Edit HEC-1 Parameters....10-9
Edit TR-20 Parameters....11-4
edit window
feature points/nodes/vertices....3-36
elevation....16-4
background map....3-33
DEM....4-8
grid node....17-5
mapping....16-6
pits....6-20
restoring TIN....6-21
vertex....6-6, 6-9
water surface....8-4
zones for snow melt....10-22
elevation zones....7-13
EMRL
fax....i
phone....i
enabling WMS....2-26
energy budget....10-21
english units....10-4
enhancing reports....3-39
equation
Manning's....15-34
weir....15-35
equations
combining from arcs....15-32
custom defined for NFF....14-5
FHWA....15-28
Maricopa County....15-30
open channel flow....15-27
shallow concentrated flow....15-27
sheet flow....15-26
TR-55....15-26
user-defined....15-31
equations for time of concentration....15-25
ESC key
creating triangles....6-3
drawing....**2-34**
select by polygon....2-27
selecting vertex strings....6-2
exit....2-26
exponential loss....10-16
export....2-20
ARC/INFO Grid....2-21
ARC/INFO point generate file....2-21
ARC/INFO TIN....2-21
basin attributes....2-21
data set....16-3
DLG....2-22
DXF....2-22
gage....16-10
HEC-1 file....10-34
stage....8-6
TIFF....2-22
TR-20 file....11-16
WKS....16-12
xy series....21-2
extrapolation....18-5
default value....18-5
inverse distance weighted....18-6
natural neighbor....18-18
false dam....6-23
feature objects
arcs....3-4
clean....3-27
commands....3-26

coverages 3-5
create arc 3-8, 5-3
creating finite difference grids 3-3
creating TINs 3-3
creating watershed models 3-2
definition 3-2
deleting 3-35
display options 3-25
nodes 3-4
points 3-4
polygons 3-5
types 3-3
vertices 3-4
feature points
importing 3-35
feature points/nodes/vertices
editing 3-36
FHWA
travel time equations 15-28
field data 16-9
file
2D grid 20-8
2D scatter point 16-3, 18-2, 20-10
ARC/INFO grid 20-20
AutoCad stream 20-26
binary 16-3
cards 20-1
CASC2D map 20-20
CASC2D project 19-2
CASC2D soils table 20-21
custom nff equation 20-34
data set 16-3
data set (ASCII) 20-12
data set (binary) 20-16
defaults 2-10
DEM 2-8, 20-5
dimensionless hydrograph 20-33
film loop 16-7
formats 20-1
gage 16-10, 20-22
GRASS 20-20
grid 2-8
hydrograph 20-31
image 2-8, 20-7
land use 20-28
map 2-8
menu 2-7
PostScript 2-25
runoff coefficient 20-30
scatter point 2-8
shape 2-10
spreadsheet 16-12
stage 20-27
stream 20-25
super 2-8, 20-2
TIN 2-8, 20-3
types 2-7, 2-9
xy series 20-32, 20-41
filling NODATA cells 4-9
film loop 16-4, 16-6–16-9
data set 16-8
dialog 16-6
display mode 16-8
file 16-7
image size 16-8
playback 16-7
saving 16-7
setup 16-7
transient 16-8
film loop animation 16-6
fit entire image 3-46
flat channel edges 6-7
flat edges 6-18
Flat objects 6-18
flat ridge edges 6-7
flat triangles 6-18
displaying 6-6
editing 6-19
large areas of 6-20
removing 6-20
flood plain
contour interval 8-8
create 8-8
delete 8-10
delineation options 8-7
description of 8-1
display options 8-2
displaying 8-3
exporting to GIS 8-9
legend 8-4
minimum 8-7
flow accumulations 5-7
flow directions 5-7
flow path 6-4
defining basins 7-10
display 7-5
downhill 7-5
maximum within a basin 15-12
uphill 7-5
flow paths 5-2, 5-11
flow patterns 7-12
display all 7-12
format
files 20-1
frame
tree 9-3

frame image 2-36
fringe....16-5
2D grid....17-5
colors....2-32
legend....2-31
ftp
USGS DEM files....4-1
gage
2D grid....17-3
computing weights....10-14
create....10-32
creating....16-9, 16-11, 17-3
curves....16-13
deleting....16-11
edit....10-32
editing....16-12
file....16-10, 20-22
import....16-10
interpolation method....16-10
plot manager....16-12
plotting....16-14
postion....10-33
printing....16-12
recording....10-34
selecting....16-11
storm total....10-34
tools....16-11
type....10-34
weights....10-14, 10-32
gages....10-32
gaging stations....7-3
get info....2-25
GIS....2-22
ASCII GRIDS....1-23
creating watershed models from....1-3
export....2-20
Importing Shapefiles....1-21
TINs....1-23
gradient....18-8
graph....16-13
GRASS
file....20-20
Green & Ampt
computing....15-1
Green Ampt loss....10-16
grid
drawing....2-35
file....2-8
module....1-25
snap to....2-35
grid generation....17-5
bias....17-6
grid node
creating....17-6
displaying....17-4
elevations....17-5
grids
creating from feature objects....3-35
HEC1
reservoirs....10-30
HEC-1....10-1
area....10-12, 11-7
basin parameters....10-11
beginning time....10-4
calibration....9-8
cards
*DIAGRAM....10-5
*LIST....10-5
all 10-9
BA....10-12, 11-7
BF....10-13
DI 10-31, 21-7
DQ....10-31, 21-7
HC....10-23
IM....10-4
IO 10-4
IT 10-4
JD....9-7, 10-5
JR 9-7, 10-7
KK....10-12, 10-24
KM....10-10
KO....10-10, 10-11
LE....10-16
LG....10-16
LH....10-17
LM....10-23
LS....10-17
LU....10-16
MA....10-22
MD....10-23
MS....10-23
MT....10-23
MW....10-23
PB....10-14
PC....21-5
PG....10-14, 10-32
PH....10-15, 21-6
PI 21-5
PR....10-14
PT....10-14
PW....10-14
QI 10-13, 10-30, 21-8
QO....10-13, 10-30
QP....10-30
RC....21-7
RK....10-29

RM 10-25, 10-29
RN 10-24
RS 10-25, 21-7
RT 10-28, 10-29
RX 21-7
RY 21-7
SA 10-27
SE 10-27, 21-7
SL 10-28
SQ 10-27, 10-28, 21-7
SS 10-28
ST 10-28
SV 10-27, 21-7
UA 21-6
UC 10-18
UD 10-20
UI 10-20, 21-7
UK 10-20
US 10-19
comments 10-10
creating without TIN 9-4
defining cross sections 21-7
depth area storms 10-5
diversion 10-30
editing 10-9
examining results 2-25
gages 10-32
general parameters 10-3
locating problems 10-35
loss 10-15
making active 9-4
multiple storms 10-7
number of ordinates 10-4
output 10-4
precipitation 10-14
printing 10-10
rainfall 10-6
Reading Existing 1-25
reading files 10-34
routing 10-23
running 10-34
snow melt 7-13
Steps to define a model 10-1
time step 10-4
writing files 10-34
help 2-6
hidden line 2-34
hidden surface 2-34
hill shade 2-34
HMS input 10-36
Holtan loss 10-17
HSPF
Reading Existing 1-27
HY8 15-40
running 15-46
saving 15-46
hydrograph
delete 9-8
display options 9-7
displaying 9-6
file 20-31
importing 9-7
input 10-30, 11-12
input for HEC-1 10-13
list 9-8
number of ordinates 10-4
observed for HEC-1 ..... 10-13, 10-30
pattern for HEC-1 10-30
printing 9-7
select 9-2
using existing 21-7
hydrologic soil group 15-1
displaying 6-7
legend 6-8
Hydrologic/Hydraulic Calculators 15-1
IDF 13-6
IDW ..... (see inverse distance weighted)
idw quadratic interpolation 6-19
image file 20-7
images 3-42
backdrop 3-43
delete 3-46
display options 3-43
importing 3-43, 3-47
mapped to TIN 3-43
reading 3-43
registering 3-44
resample 3-46
tiff 3-42
import
xy series 21-2
import
ARC/INFO TIN 2-19
attributes of shape file 2-11
DEM 2-15
DEM attributes 2-18
DXF 2-20
feature points 3-35
HEC-1 file 10-34
hydrographs 9-7
land use data 2-19
soil type 2-19
stage 8-4
tabular data 2-19
TIFF 2-20
TR-20 file 11-16

XYZ data 2-20
import grid
2D 2-18
IN21-8
inactivate DEM points 4-6
Inactive DEM points 4-4
incremental rainfall 21-5
index
2D scatter point 18-3
cell 17-3, 17-5
index of storms....9-8
infiltration 16-9
interpolate
DEM to TIN 4-8
interpolation
cluster 18-9
global....18-10
gradient....18-8
IDW..... (see inverse distance weighted)
inverse distance weighted(see inverse distance weighted)
linear....18-5
local 18-10, 18-13
local coordinates....18-16
natural neighbor..... (see natural neighbor)
nodal functions 18-8
options 18-4
quadratic 18-8
rainfall to basins 18-19
Shepard's method;.i.Shepard#(see inverse distance weighted);.i.inverse distance weighted
Shepard's method 18-6
stage.... 8-5, 8-8
stage values....8-3
subsets 18-9
to grids....18-19
truncation....18-4
vertex z values 6-10
inverse distance weighted 18-5–18-12
barycentric weights....18-11
interpolation subsets 18-9
local weighting method....18-10
local/global 18-10
nodal functions 18-8
weights....18-6
kinematic wave routing 10-29
kinematic wave unit hydrograph 10-20
labeling geometric attributes....5-4
lag time 10-19, 10-20
Colorado State equation 15-20
computing....15-5
Denver equation....15-15
Eagleson equation 15-18
Espey equation 15-16
general equation 15-14
Putnam equation....15-19
SCS equation....15-21
Taylor and Schwartz equation....15-19
Tulsa district equation 15-14
land use.... 3-16, 4-4, 6-8, 15-1
file 20-28
legend....6-8
lat/lon
converting to UTM....4-9
legend 2-31
flood plain 8-4
hydrologic soil group 6-8
land use 6-8
length
TR-20 reach 11-10
length ratio....6-18
lighting....2-34
links 3-38
local minima 6-7
locked vertices....6-11, 6-20
logarithmic contours 2-31
loss
channel 10-29
exponential 10-16
Green Ampt....10-16
Holtan....10-17
SCS 10-17
snow melt 10-23
uniform....10-16
macro....2-6
Manning's equation 15-34
manual redraw 2-35
map
file 2-8
module....1-24
map module
overview....3-1
mapping.... (see elevation)
Maricopa County
travel time compuations 15-30
mean 16-4
menu
bar 2-7
data....16-1
display....2-28
edit 2-26
interpolation 18-4
view....2-36
metric units 10-4
minimum flood plain depth....8-7

model check 10-35
TR-20 11-17
model checker
CASC2D 19-15
MODRAT 13-17
module 1-23
DEM 1-24
design of 2-1
grid 1-25
map 1-24
scatter point 1-25
selecting a new 2-3
TIN 1-23, 6-1
tree 1-24
mosaic
USGS file transfer 4-1
move to back 3-42
move to front 3-42
MS Windows
printing 2-25
multiple selection 2-6
arcs 3-9, 5-3
Muskingum routing 10-25
Muskingum-Cunge routing 10-29
National Flood Frequency 14-1
natural neighbor 18-14–18-19
bounding window 18-18
extrapolation 18-18
local coordinates 18-16
weights 18-17
new 2-8
NFF 14-1
computing peak discharge 14-5
defining your own equation 14-5
input dialog 14-2
saving 14-6
NO DATA 4-4
nodal function 18-8
node
edit 3-7, 5-2
grid ..... (see grid node)
nodes 3-4
converting to vertices 3-26
edit 3-7, 5-2
snapping together 3-27
null basin triangles 7-11
number of ordinates 10-4
oblique ..... (see viewing)
open 2-8
open channel flow 15-27
outlets 6-7
creating 7-6
creating for trees 9-4
default 7-6
definition of 7-6
delete 7-6
deleting from trees 9-5
displaying 7-3
inserting in trees 9-4
names 7-3, 10-24, 11-10
select 9-2
tree 9-1
overland flow 19-12
overland flow distance 15-12
overview 1-1
page layout 2-23
palette
module 2-3
pan 2-4
patch
Clough-Tocher 18-13
pipes 7-8
pits 6-18
displaying 6-7
editing automatically 6-20
smooting 6-20
pixel map 16-6
plan view ..... (see viewing)
plot options ..... (see gages)
point
edit 3-7, 5-2
point/node type
CASC2D 3-22
drainage 3-12
points 3-4
delete 3-32
edit 3-7, 5-2
polygon 2-27
activate region 17-7
Thiessen 18-15
polygon type
CASC2D 3-24
drainage 3-13
polygons
creating from arcs 3-33
feature objects 3-5
polyline 6-16
postscript 2-22
PostScript 2-25
precipitation
TR-20 11-4
precipitation ..... (see rainfall)
CASC2D 19-8
printing 2-22
display options 2-22, 2-24
gage plots 16-12

hydrographs 9-7
page layout....2-23
PC....2-25
position on page....2-23
postscript 2-22
setup 2-22
size....2-23
UNIX....2-25
privileges
write....16-3
quit....2-26
rainfall....10-13
basin average 10-14
cumulative 21-5
defining xy series....21-5
hypothetical storm 10-15
incremental 21-5
none 10-14
Thiessen method....10-32
type....10-6
rational method
computing flows 13-4
editing basin data....13-4
editing outlet data 13-5
equation parameters....13-2
hydrographs 13-9
saving....13-16
Rational Method 13-1
REACH....11-9
read
TR-20 file 11-16
reclassification 19-11
recording gage....10-34
redistributing vertices.... 3-28, 3-33
redistribution
along a spline....3-29
uniformly 3-29
references....21-1
refined basin boundaries 7-10
refresh 2-34
refreshing the display....2-35
region
activate with polygon....17-7
regional regression equations....14-3
register 2-26
registering images 3-44
removing
duplicate vertices....6-12
flat triangles....6-20
rendering....(see shading)
renumbering links and nodes....3-38
reorder stream arcs....3-28
resample....3-46
reservoir
creating....7-9
defining in TR-20....11-14
definition of....7-8
delete....7-10
storage capacity curves from TINs....7-9
reservoir analysis....15-36
reservoirs
adding to trees....9-5
deleting from trees....9-5, 9-6
restore elevations....6-21
retriangulation....6-10
RETURN key
selecting vertex strings....6-2
reverse arc direction 3-33
ridge edges....6-6
ridges
node location....3-27
rotation 2-4
routed hydrographs....10-10
routing
channel 10-26
kinematic wave....10-29
Muskingum 10-25
Muskingum-Cunge....10-29
none....10-24
parameters 10-23
reservoir 10-27
storage....10-25
Straddler/Stagger....10-28
TR-20 ....11-9, 11-10
running HEC-1 10-34
running TR-20 11-16
RUNOFF 11-6
runoff coefficient
file 20-30
runoff coefficient table 15-5
runoff coefficients....15-1
save as 2-9
save defaults 2-10
saving DEM elevations....4-8
scaling vertices 6-12
scatter point 1-25
file 2-8
from TIN 6-24
screen layout....2-1
SCS
loss 10-17
unit hydrograph 10-20
SCS Curve Number
TR-20 11-7
select....2-5
2D scatter point 18-2

2D scatter point set 18-2
all 2-27
arc 3-7, 5-2
arc vertices 3-7, 5-2
basins 6-3, 9-2, 10-2, 11-2
boundary triangles 6-17
branch 3-9, 5-3
cell 17-2
DEM point 4-2
diversion 9-2
gage 16-11, 17-4
grid column 17-3
grid row 17-3
hydrograph 9-2
multiple 2-27
multiple objects 2-6
network 3-9, 5-3
nodes 3-7, 5-2
outlets 10-2, 11-2
points 3-7, 5-2
SHIFT key 2-5
triangles 6-2
vertex string 6-2
vertices 6-1
with poly 2-27
xy series point 21-4
set active region 4-5
settings 2-10
shading **2-34**
color fill contours 2-31
command 2-35
flat 2-35
fringes 16-5
lighting 2-34
options 2-34
smooth 2-34, 2-35
shallow concentrated flow 15-27
shape files
ARC/INFO files 2-10
Shapefiles
Importing 1-21
sheet flow 15-26
SHIFT key
multiple selections 2-5
zooming 2-4
zooming in xy series 21-5
shortwave radiation 10-23
shuffle down 3-42
shuffle up 3-42
slope 15-12
smoothing 6-14
DEMs 4-6
pits 6-20
TINs 6-20
smoothing CASC2D stream cells 3-37
snap to grid 2-35
snapping streams/ridges 3-27
snow melt 10-21
dew point 10-23
elevation zones 10-22
loss 10-23
shortwave radiation 10-23
temperature 10-23
wind speed 10-23
Snyder unit hydrograph 10-19
soil type 4-4
soils 3-17
spline 2-31, 3-29
split flow vertices
correcting 7-11
displaying 6-7
spreadsheet 16-12
spreadsheet data 2-21
stage
creating streams for 8-6
definition of 8-4
deleting 8-5
displaying 8-3
editing 8-5
file 20-27
fixed 8-3
importing 8-4
interpolation 8-3, 8-5, 8-8
saving 8-6
values 8-3
standard deviation 16-4
starting time 11-3
storage capacity curves 7-9
storage coefficient 15-23
storage routing 10-25
storm index 9-8
storm total gage 10-34
Straddler/Stagger routing 10-28
streams
branching 7-6
creating a pipe 7-8
creating from arcs 3-33
creating from triangle edges 7-7
definition of 7-7
delete all 7-8
deleting 7-8
displaying 7-3
displaying profile 7-8
editing 7-8
file 20-25
importing digitized 2-14

length 15-13
node location 3-27
reorder arcs 3-28
slope 15-13
stage 8-6
urban 7-7
string
vertex 6-2
subdivide 6-14
subdivision 6-14
subdivision factor 6-19
super file
WMS 2-8, 2-9, 20-2
surface dpeth 16-9
swap edge
correcting split flow vertices 7-11
swap edges 6-3
automatically 6-23
symbol
2D scatter point 18-3
table of contents i
tabular data 2-19
TAPE22 file 9-6, 20-31
temperature 10-23
Thiessen polygon 10-32, 18-15
TIFF
export 2-22
import 2-20
tiff images 3-42
time increment 11-3
time of concentration 3-18, 10-19, 11-8, 12-4, 15-24
computing 15-5
Fort Bend County equation 15-23
Kerby equation 15-22
Kirpich equation 15-22
Ramser equation 15-23
time step 10-4
TIN 6-1
boundary 6-10, 6-17
breakline 6-16
converting to DEM 6-24
converting to feature objects 7-13
converting to feature polygon 6-23
convex hull 6-17
creating from DEMs 3-33
creating from feature objects 3-33
creating triangles 6-3
creating vertices 6-2
delineating watersheds 1-15
display options 6-4
displaying boundary 6-5
editing 6-23, 7-1
editing automatically 6-18
file 2-8, 20-3
filter options 6-21
filtering 6-20
from 2D grid 17-8
Importing from GIS 1-23
merging 6-15
module 1-23, 6-1
smoothing 6-14, 6-20
storage capacity curves 7-9
subdividing triangles 6-14
z value 6-10
tool
CASC2D dynamic palette 19-2
create arc vertex 3-8, 5-3
create ellispse 3-39
create line 3-39
create points 3-8, 5-3
create rectangle 3-39
create text 3-39
create triangles 6-3
create vertices 6-2
create xy series point 21-5
DEM contour labels 4-2
DEM dynamic palette 4-1
drawing objects palette 3-39
dynamic palette 2-5
feature objects palette 3-7
flow path 5-2, 6-4
palette 2-3
pan 2-4
place basin labels 6-3
place drainage label 5-4
rotate 2-4
select 2-5
select arc 3-7, 5-2
select arc vertices 3-7, 5-2
select basins 6-3, 9-2
select branch 3-9, 5-3
select dem point 4-2
select diversion 9-2
select drawing objects 3-40
select hydrograph 9-2
select network 3-9, 5-3
select nodes 3-7, 5-2
select outlets 9-2
select points 3-7, 5-2
select polygon 3-9, 5-3
select triangles 6-2
select vertex strings 6-2
select vertices 6-1
select xy series point 21-4
static palette 2-4

swap edges 6-3
TIN contour labels 6-3
TIN dynamic palette 6-1
topologic tree dynamic palette 9-2
zoom 2-4
zoom xy seriex 21-5
TOPAZ 5-7, 5-12
TR-20 11-1
ADDHYD 11-9
base flow 11-4
Basin Geometric Attributes 11-8
basin name 11-7
basin parameters 11-6
creating without TIN 9-4
discharge hydrographs 11-6
diversion 11-14
editing parameters 11-4
general parameters 11-2
input hydrographs 11-12
locating 11-17
making active 9-4
output 11-5
precipitation 11-4
reach length 11-10
Reading Existing 1-26
reading files 11-16
reservoir 11-14
routing methods 11-10
routing names 11-10
routing parameters 11-9
running 11-16
RUNOFF parameters 11-6
starting time 11-3
Steps to define a model 11-1
time increment 11-3
time of concentration 11-8
title 11-3
unit hydrograph 11-3
Use Defined Cross Section 11-8
Use Defined Reservoir 11-9
writing files 11-16
TR-55 12-1
basin data 12-2
hydrographs 12-5
input requirements 12-1
outlet data 12-2
time of concentration 12-4
time of concentration equations 15-26
transient
animation 16-8
translating nodes 3-27
travel time 15-24
travel times
combining arcs within a basin 15-32
tree
basin creation 9-5
centering in window 9-3
creating 9-4
delete outlets 9-5
diagram 10-5
display options 9-2
displaying 9-3
diversion creation 9-5
insertion of outlets 9-4
module 1-24
reservoir creation 9-5
saving HEC-1 file 10-34
triangle
boundary 6-17
create 6-3
displaying 6-5
edges 6-3
edit 6-2
in multiple basins 7-11
length ratio 6-18
long/thin 6-17
select 6-2
triangulation 6-3, 6-12
command 6-14
convex hull method 6-13
creating a TIN from vertices 6-14
Delauney 6-6, 6-16, 18-15
display of 6-13
enclosing triangle method 6-13
enclosing triangle plus fill method 6-13
options 6-13
retriangulate 6-10
TINs from feature objects and DEM 3-33
truncation
interpolation 18-4
uniform loss 10-16
unit hydrograph 10-18
Clark 10-18
Dekalb 13-15
given 10-20
kinematic wave 10-20
rational 13-12
SCS 10-20
Snyder 10-19
universal rational 13-14
user-defined 13-16
units 6-12, 7-12, 10-4
UNIX
printing 2-25
unlocked vertices 6-11, 6-20
urban stream networks 7-7

USGS 4-1, 6-20
vertex
creating 6-2, 6-8
deleting 6-9
displaying 6-5
editing 6-1, 6-9
elevations 6-6, 6-9
lock/unlock 6-11
options 6-8, 6-10
remove duplicates 6-12
select 6-1
string 6-2
vertex string 7-7
vertices
converting to nodes 3-26
density 3-29
feature objects 3-4
redistribution 3-28
view
angle 2-36, 2-37
bearing 2-36
dip 2-36
file 2-25
frame image 2-36
last 2-37
oblique 2-37
pan 2-4
plan 2-37
rotate 2-4, 2-37
window limits 2-36
z magnification 2-36
zoom 2-4
watershed 7-3
calculating cumulative areas 7-12
color 7-5
displaying 7-12
watershed delineation from DEMs 5-1
watershed models
using DEMs 1-6
using feature objects 1-3
using GIS data 1-3
using TINs 1-15
watershed storage coefficient 15-23
weir calculations 15-35
wind 10-23
window
bounds 2-36
edit 2-6, 3-7, 5-2, 6-1
film loop 16-8
gage plot 16-12
graphics 2-2
help 2-6
hydrograph 2-2
tool palette 2-3
wireframe 2-24
WKS *16-12*
WMS tree file 9-9
world file 3-45
write
TR-20 file 11-16
xy series
interpolate
xy series 21-4
active 21-2
compress 21-4
creating point 21-5
cross sections 21-7
cumulative rainfall 21-5
defining rainfall 21-5
delete 21-2, 21-3
dialog 21-3
displaying 21-4
diversion data 21-7
duplicate 21-2
editing 21-3
editor 21-1
export 21-2
file 20-32, 20-41
frame 21-5
given hydrograph 21-8
hypothetical storm 21-6
import 21-2
incremental rainfall 21-5
list 21-2
pan 21-5
plot 21-4
selecting point 21-4
standard storm 10-7
storage routing curves 21-7
time area curve 21-6
update 21-4
zoom 21-5
XYZ data
import 2-20
z magnification 2-36
z value 6-8, 6-10
confirm 6-10
extrapolate 6-10
interpolate 6-10
zoom 2-4